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Preface

This document is a user's guide for the Intergraph Smart[™] 3D Drawings and Reports task and provides command reference information and procedural instructions.

The Drawings and Reports user documentation is delivered in three separate documents:

- Orthographic Drawings User's Guide
- Isogen Isometric Drawings User's Guide
- Reports User's Guide

Documentation Comments

For the latest support information for this product, comments or suggestions about this documentation, and documentation updates for supported software versions, please visit Intergraph Smart Support (https://smartsupport.intergraph.com).

ISO Standards in Marine Drawings

Intergraph SmartTM 3D marine drawings are ISO compliant. Several ISO templates are delivered with the product.

Below is a list of ISO Reference numbers and the description of the document.

ISO 128-20:1996(E)	Technical Drawings – General principles of presentation Basic conventions for lines
ISO 128-21:1997(E)	Technical Drawings – General principles of presentation Preparation of lines by CAD systems
ISO 128-22:1999(E)	Technical Drawings – General principles of presentation Basic conventions for applications for leader lines and reference lines
ISO 128-23:1999(E)	Technical Drawings – General principles of presentation Lines on construction drawings
ISO 128-24:1999(E)	Technical Drawings General principles of presentation Lines on mechanical engineering drawings
ISO 128-25:1999(E)	Technical Drawings General principles of presentation – Part 25: Lines on shipbuilding drawings
ISO 128-30:2001(E)	Technical Drawings – General principles of presentation – Part 30: Basic conventions for views
ISO 128-40:2001(E)	Technical Drawings – General principles of presentation – Part 40: Basic conventions for cuts and sections

ISO 128-50:2001(E)	Technical Drawings – General principles of presentation – Part 50: Basic conventions for representing areas on cuts and sections
ISO 5457:1999(E)	Technical product documentation – Sizes and layout of drawing sheets
ISO 7200:2400(E)	Technical product documentation – Data fields in title blocks and document headers
ISO 5261:1995(E)	Technical Drawings – Simplified representation of bars and profile sections
ISO 2553:1992(E)	Welded, brazed and soldered joints – Symbolic representation on drawings
ISO 5572:1987(E)	Shipbuilding and marine structures – Numbering of equipment and structural elements in ships
ISO 7462:1985(E)	Shipbuilding – Principal ship dimensions. Terminology and definitions for computer applications
ISO 8193:1984(E)	Shipbuilding – Shell plating information
ISO 9203-1:1989(E)	Shipbuilding – Topology of ship hull structure elements – Part 1: Location of elements
ISO 9203-2:1989(E)	Shipbuilding – Topology of ship hull structure elements – Part 2: Description of elements
ISO 9203-3:1989(E)	Shipbuilding – Topology of ship hull structure elements – Part 3: Relations of elements
ISO 129-1:2004(E)	Technical drawings – Indication of dimensions and tolerances – Part 1: General principles
ISO 6428:1982(E)	Requirements for microcopying
ISO 3898:1997(E)	Bases for design of structure – Notations – General symbols
ISO 3098-0:1997(E)	Technical product documentation – Lettering – Part 0: General requirements
ISO 3098-2:2000(E)	Technical product documentation – Lettering – Part 2: Latin alphabet, numerals and marks
ISO 3098-5:1997(E)	Technical product documentation – Lettering – Part 5: CAD lettering of Latin alphabet, numerals and marks

By default, the following metric drawing templates are delivered in the ISO format.

LARGE

- Paper Format (ISO 5457)
 - A0 (valid for A1 and enlarged formats)
 - Title-block (ISO 7200)
- Lines (ISO 128 Part.25, ISO 6428)
 - Wide 0.7 mm
 - Narrow 0.35 mm
- Characters (ISO 3098-5, ISO 6428)
 - Titles 7 mm (ISO 3098 BVL 7)
 - Labels 3.5, bold or 5 (ISO 128 Part.22)
 - Dimensions 3.5, values rounded to 1 mm (ISO 129)
- Symbols to match above lines and characters for
 - Relative positions of structural elements (BL, CL,...)
 - Sections and details (Section plane, Section and Detail Titles)
 - Profile cross-sections (ISO 5261)
 - Profile end-cuts re-symbolization
 - Section and plates butt joints
 - Welds (ISO 2553)

What's New in Drawings and Reports

The following changes have been made to the Drawing and Reports task.

Version 2016 (11.0)

Drawings (General)

- You can now transfer the ownership of drawings between permission groups. For more information, see Configuration Tab (Properties Dialog Box) (on page 69). (P1 CP:278233)
- A new 3D Dimension ribbon in SmartSketch Drawing Editor overrides manual 2D dimensions with 3D dimensions. For more information, see *Dimensioning in 3D Drawings* (on page 337) and 3D Dimension Ribbon. (P1 CP:139408)
- You can copy a drawing view only by using the Copy and Paste View command. For more information, see Copy and Paste View Command (on page 458). (P2 CP:226825)
- You can perform Update Selected on both full and cropped views. For more information, see Drawings View Explorer (on page 324). (P2 CP:230723)
- Added a new ribbon, Generic View Ribbon, to the Associate Objects to View command that allows you to specify options on a selected graphic view with a generic rule set view style. For more information, see Associate Objects to View Command (on page 374). (P2 CP:244388)

- You can now customize the shortcut menu on a folder in the Management Console. For more information, see New Command (on page 38). (P2 CP:255695)
- You can now either delete a manual view placed by rule, or move the view to the UnAssigned Folder. For more information, see *Delete Views* (on page 507). (P2 CP:256645)
- You can now customize Drawings by Rule Steel Order rulesets in .NET to generate drawings. For more information, see Drawings by Rule Components. (P2 CP:266707)
- In drawings by rule, view annotations, such as the ruler and view name, are automatically adjusted in relation to the **View Offset** value. For more information, see *View Tab (Drawing View Properties Dialog Box Drawings View Explorer)* (on page 355). (P2 CP:267889)
- Smart 3D displays a warning telling you to use **Copy and Paste View** arther than the traditional **Cut**, **Copy**, and **Paste** commands with views. For more information, see *Copy and Paste View Command* (on page 458). (P2 CP:226825, CP:275427, and CP:284427)
- You can now prevent landing curves from occluding other landing curves using the AddSolidForVHL flag. For more information, see Generic Stiffener Landing Curve. (P2 CP:296036)
- Group Selected Labels stacks selected objects without overlapping the leader lines. For more information, see *Group Selected Labels* (on page 419). (P2 CP:265171 and P3 CP:271343)
- You can select multiple objects with the Place a Label command, and place a group label for the objects. For more information, see Place a Label Command (on page 400) and Group Labels (on page 405). (P3 CP:181864)
- Drawing by Rule view properties only display applicable drawing by rule view styles. For more information, see *View Tab (Drawing View Properties Dialog Box - Drawings View Explorer)* (on page 355). (P3 CP:215229)
- You can place a manual view for non-shell plates in Marine mode. For more information, see Place a Manual View for Non-Shell Plates (on page 366).
- You can now update all drawings by rule documents with the Full Update action in Intergraph Smart Batch Services. For more information, see Full Update Action. (P1 CP:160700) (P3 CP:199390)
- A new property, View Cone Angle, displays the cone angle value. For more information, see View Tab (Drawing View Properties Dialog Box - Drawings View Explorer) (on page 355). (P3 CP:266696)
- Added a new Warning view state to the Drawings View Explorer. For more information, see Drawings View Explorer (on page 324). (P3 CP:271093)
- In ruleset views, if you change the view style before editing View Offset for the first time,
 View Offset displays the offset value as defined in the .xml template of the view style. For more information, see View Tab (Drawing View Properties Dialog Box) (on page 349). (P3 CP:272849)
- A new file option, File Already Exists Action, specifies how to save the hierarchy if a file
 with the same name already exists. For more information, see Save As Command (on page
 72) (P4 CP:273965)

- Any time you update a 3D model document, the software now generates a single log file containing status information and any errors encountered during the process. The log file location is %TEMP%\EFUpdateCache\[3D Model ID]\[3D Model ID]\[3D Model ID]\]. For more information, see *Updating Documents* (on page 82). (P2 CP:292733)
- Added information about hiding plant view styles when working in marine mode. For more information, see View Tab (Drawing View Properties Dialog Box - Drawings View Explorer) (on page 355). (P3 CP:293501)

Orthographic Drawings

- You can now export 3D model graphics directly to MicroStation V8 DGN file format. For more information, see Exporting 3D Model Graphics to MicroStation. (P4 CP:251155)
- Added a new MicroStation Version option to Setup dialog box (MicroStation DGN Files).
 For more information, see Setup Dialog Box (MicroStation DGN Files). (P4 CP:112450)
- Added information about the working units supported by the seed file when exporting graphics to MicroStation V7 DGN file format. For more information, see Create MicroStation DGN files .(P2 CP:75204)
- Highlight Annotations and Clear Manual Edits commands are available on the Drawings Compose toolbar. For more information, see *Highlight Annotations Command* (on page 460) and *Clear Manual Edits Command* (on page 462). (P2 CP:256406)
- You can now associate manually placed objects to graphic views. For more information, see Associate Graphics to Graphic View Command (on page 463). (P2 CP:247816)

Isogen Isometric Drawings

- You can now produce cable tray and HVAC isometric drawings. To support this enhancement, the Style list on the Setup dialog box now enables you to filter styles according to the isometric drawing type. For more information, see Setup Dialog Box (Isogen Isometric Drawing by Query Component) (on page 106).(P2 CP:273785 and CP:209035)
 - ★ IMPORTANT In this version of the software, cable tray and HVAC isometric drawing options are limited to beta access. As such, isometric options specific to cable tray and HVAC drawings are not visible in **Isogen Configuration** by default. To turn on cable tray and HVAC options so that you have access to the full array of isometric drawing settings provided by the software, please contact *Intergraph Support* (http://www.intergraph.com/support).
- To support the creation of cable tray and HVAC isometric drawings, two new drawing styles
 are delivered with the software: Iso_CableTray and Iso_HVAC. For more information, see
 Isometric Drawing Styles (on page 125).
- The function of the piping component file (PCF) has been extended. In addition to piping content, the PCF now supports the transfer of HVAC and cable tray content and configuration information between Smart 3D and Isogen.
- You can now associate manually placed objects to graphic views. For more information, see Associate Graphics to Graphic View Command (on page 463). (P2 CP:247816)
- In previous versions of the software, you used **TreatTapsOnPipeAsTeeStubs** to specify whether the software managed taps as tee stubs or as regular taps when generating a piping isometric drawing or when creating a piping component file (PCF). The software now treats all radial taps placed on pipe as set-on tees, or tee stubs. All non-radial taps remain as taps. (P2 CP:255192)

- Previous versions of Smart 3D refer to Piping Isometric Drawings. Because the software now supports the production of cable tray and HVAC isometric drawings, the UI now reflects the generic term Isogen Isometric Drawings. This document has been updated to reflect this change in the UI. (P2 CP:259784)
- The **Isometric Style Options Browser**, which enabled you to control all of the options related to the appearance and information content of the isometric drawing, has been replaced with **Isogen Configuration**. For more information, see *Isogen Configuration* (on page 136). (P2 CP:255749)
- You can now show the slope symbol without any angle/ratio for sloped lines. (P3 CP:157751)
- Isogen Configuration supports the ability to import a style fragment. For more information, see Import data from a style XML file (on page 173). (P2 AL:14940)
- You can export a customized set of isometric drawing properties, Alternative Text, and report definitions to a style .xml file. For more information, see Export an isometric drawing style fragment (on page 176) and Export Style Fragment Dialog Box (on page 177). (P2 AL:14965)
- Isogen Configuration supports a new file path macro, \$SYMBOLSHARE\$. This macro
 enables you to reference the SharedContent folder when specifying any path within an
 isometric drawing style. Previous versions of the software required you to specify a hardcoded path to reference any file located in the SharedContent folder. (P2 CP:105056)
- PCF syntax now supports the following material attributes: GEOMETRIC-STANDARD, MATERIAL-OF-CONSTRUCTION, OUTSIDE-DIAMETER, OUTSIDE-DIAMETER2, RATING, RATING2, SCHEDULE, SCHEDULE2, WALL-THICKNESS, and WALL-THICKNESS2. (P2 CP:248316)
- A new PDF document is delivered with the software. The Isometric Drawing Options Reference Guide contains descriptions and general information for working with the Isogen and Smart 3D isometric drawing properties. In previous versions of the software, this content was delivered as part of the Piping Isometric Drawings User's Guide. The new document (IsometricDrawingOptionsReferenceGuide.pdf) is available in the [Product Folder]:\Documentation\Help folder and from the Help > Printable Guides command in the software.
- Added guidelines for creating a dotted dimensioned label. For more information, see *Create a dotted dimensioned label* (on page 182).
- The following changes are specific to the isometric drawing properties supported by the software. For more information, see What's New in the Isometric Drawing Options Reference Guide. You can access this document using the Help > Printable Guides command in the software.
 - A new options group, Grouping, has been added to **Drawing Area > Graphics**. (P2 AL:10443)
 - The **Group graphics by UCI** property allows you to control how graphics are grouped in the output CAD file. (P2 AL:10443)
 - The Welds and Joints style category has been re-organized to improve its overall layout. (P2 AL:14803)
 - The software now supports horizontal positioning of the dimension standout. (P2 AL:11970)

- The software supports the ability to pull properties directly from the Drawing Sheet object. (P2 CP:123112)
- The number of user-definable Alternative Text (AText) strings that the software supports has been expanded. (P2 CP:74293)
- A new OidType called Drawing Sheet enables you to pull properties directly from the Drawing Sheet object.(P2 CP:123112)
- The software supports output of a double-banked material list on the isometric drawing. Previous versions of the software supported offsetting the material list section only along the Y (or vertical) direction of the drawing using Section2 Y Offset and Section3 Y Offset. Now you can also offset the material list section along the X (or horizontal) direction of the drawing using Section2 X Offset and Section3 X Offset. You can set this option using Materials > Drawing Material List > User Defined. (P3 CP:250733)
 - **NOTE** You can also use the **Drawing Setup Tool** to set the horizontal offset for the report data contained in Section 2 and Section 3 of the material list. For more information, see **X** under **Section Two** and **Section Three** in *Drawing Areas Page* (on page 249) and *Material List Page* (on page 253).
- The software supports user-specified reference dimension placement locations on a piping isometric drawing. As part of this new functionality, the **Placement** property is renamed to **Automatic Placement**, and a new setting, **Off**, has been added. (P3 CP:176166)
- Added information about using pipeline data text attributes to dynamically name Isogen output files.

Reports

- You can now specify a unit delimiter between primary and secondary units in a report. For more information, see Parameters (Report Shortcut Menu). (P2 CP:140982)
- Added a new report, Designed Member Itemized Material Take-off. For more information, see Designed Member Itemized Material Take-off. (P2 CP:234498)
- Added the trench run report. For more information, see Trench Run Report (Civil). (P2 CP:259491)
- Added a new report, Diagnostic Synch Workspace Report (Diagnostic). For more information, see Diagnostic Synch Workspace Report (Diagnostic). (P2 CP:269935)
- Added information on using the Reporting Requirement property to exclude objects from reports. For more information, see Report Queries to Extract Data. (P3 CP:85240)
- Added the workflow for configuring and running personal reports. For more information, see Run Report. (P4 CP:281013)

SECTION 1

Drawings and Reports

The Intergraph Smart[™] 3D Drawings and Reports task creates orthographic drawings (by a variety of methods), isometric drawings, and reports from the model. When the 3D model changes, you can update your drawings and reports.

The Drawings and Reports task is also responsible for publishing your drawings and reports. When your model is registered using the SmartPlant Registration Wizard, you can publish volume and composed drawings, orthographic drawings, isometric drawings, and reports. You can also publish 3D model data using the 3D Model Data component.

The **Management Console** organizes the different document types into a customizable hierarchy. Using the component functionality of the console, you can create, edit, update, print, save, and publish the deliverables. This hierarchy of components and documents is also available in many of the 3D tasks, such as Common, by using the **Tools > Drawing Console** command. You can perform nearly all of your document operations using this command. For more information, see the *Common User's Guide*.

Before you can create components for drawings and reports, your administrator must organize the **Management Console** hierarchy with folders for each component type. Then, the administrator must complete several setup steps, including setting up drawing and report templates, creating view styles, creating appropriate filters, and specifying isometric drawing options. Default templates and view styles are delivered with the software, and you can customize them to suit your needs.

It is possible to customize templates and view styles before any objects exist in the model. However, to create drawings and reports, objects must exist in your model. For example, if you want to generate Isogen isometric drawings, you must have piping, cable tray, or HVAC data in your model.

Composed Drawings

Composed drawings are orthographic drawings created in a 3D task such as Common. The composed drawing component, available in the Drawing and Reports task **Management Console**, manages the composed drawings you create. Composed drawings are flexible, allowing you to have views that are managed by a drawing region and associate the views to volumes and other views.

Volume Drawings

Volume drawings are useful for creating general arrangement or construction drawings of areas within the model. In the Volume Drawing workflows, you or your administrator must create or edit border templates. You can place drawing property labels in the title block of the template to fit your company or project. You also must configure the view styles, which are sets of rules that determine how the graphics in the three-dimensional model are represented on the drawings. View styles use filters. You can create a folder of drawing filters, with new, existing, and future filters for each discipline. You place drawing volumes in the Space Management task. You can publish Volume drawings when they are up-to-date.

Orthographic Drawings by Query

The Orthographic Drawing by Query component, in conjunction with the Drawings by Query Manager component, creates drawings in mass by specifying a filter-based query to collect objects for drawings. This drawing type is appropriate for creating detail drawings of particular objects within the model. They are especially useful when creating drawings that use the same style or format for large numbers of similar objects, such as hangers or supports.

Just like composed and volume drawings, you can print, update, save into MicroStation J (V7 and V8) or AutoCAD formats, or publish Orthographic Drawings by Query. When you publish Orthographic Drawings, a viewable graphic file is created; no physical data is published.

Orthographic Drawings by Rule (marine mode and material handling mode)

Drawing by Rule types include design and detail drawings, manufacturing drawings, and assembly drawings. You can use the Drawings by Rule component to create custom drawings, or select a delivered Drawings by Rule package to create your drawings.

Isogen Isometric Drawings by Query

Like Orthographic Drawings by Query, you create Isogen Isometric Drawings by Query by specifying a filter-based query to collect the objects. The workflow requires that you create or edit border templates to fit your company or project. You or your administrator also must set the isometric options for each of the isometric styles that you need in your project.

When you publish Isogen Isometric Drawings by Query, they are published as viewable graphics. No physical data is published.

Reports

In the Spreadsheet Reports workflow, you create report templates, which control the content and format of reports. The default file format of reports in the software is Microsoft Excel® format. The Report Template Editor provides the ability to configure your reports to use queries and special formatting.

You can publish Spreadsheet Reports just like drawings. However, the Spreadsheet Reports are published as Excel spreadsheets; no physical data is published.

■ NOTES

- Microsoft Excel 2003 is the minimum supported version for the Drawings and Reports task.
- Microsoft Excel 2007 has a file format and extension of .xlsx. However, the delivered report templates still use the old .xls extension. If you create a report template with the .xlsx extension, only machines with Excel 2007 will be able to handle (edit, update, or open) the report. If you attempt to open an .xlsx file with an older version of Excel, an error message displays stating that the file is not compatible with the version of Excel.
- If you use Office 2003, in Microsoft Excel under Tools > Macro > Security > Trusted
 Publishers tab, check the Trust Access to Visual Basic Project option.
- If you use Office 2007 and Office 2010, click the Microsoft Office button to open Excel Option. Go to the Trust Center category and select the Trust Center Settings button. Select the Macro Settings category and check Trust access to the VBA project object model.
- For more information about Microsoft Office and service packs, refer to the Microsoft web site (http://www.microsoft.com/) (http://www.microsoft.com/).

SECTION 2

Delivered Drawing Types

The delivered drawing types are drawing templates that include drawing borders, documentation annotation, note areas, and selection and resymbolization criteria.

Several drawing types are delivered and fully designed to meet particular drawing requirements. You can use the delivered types to create new drawing types and to modify the view styles or border templates as needed. You can copy a template from an existing drawing or you can copy volumes only, allowing you to create multiple drawings with the same border graphics. To copy a drawing type component, select the item on the **Management Console** hierarchy, and then select **Copy**. To paste the item, right-click a location in the hierarchy or in the **Detail View** and select **Paste**.

All of the delivered drawing types provide customizable templates and view styles. The delivered **Equipment Plan** drawing type is provided as an example below:

- The **Equipment Plan** is a single view drawing plan. It includes the location of equipment, structural columns, building walls, equipment steel, vessel and mechanical steel, and roads.
- The Equipment Plan includes general information for coordinate systems, sheet scales, and modifications. The drawing border provides the border graphics, title block graphics, and the title block labels.

The document annotation includes the following:

- North Arrow Indicates the orientation of the model. The large symbol is used, which is
 typical for single view drawings. The north arrow is placed on each drawing view. Click and
 drag the symbol to position it within the Note Area if required.
- Key Plan Shows the geographic position of the single grid relative to the rest of the grids of the same type in the single block.
- Drawing Notes Shows the collection of notes, which consists of general notes applicable
 to all drawings, notes applicable to a discipline, notes applicable to a category of drawings,
 and notes specific to a single drawing (such as an element or a border report).

Note Area

The Note Area is used to display drawing notes and key plans. The note area on the Equipment Plan is five inches on the right-hand side of the drawing border. This area extends from the top of the border down to the top of the title area border. The Note Area is optional, and is not a required element in the template.

View Regions

The View Region defines the drawing view arrangements. The **Equipment Plan** defaults to a single view with a five-inch Note Area and one-inch margins around the drawing view.

Drawing View

The **Equipment Plan** is a single view plan. The following specifications are set:

- Direction Set to Plan view (Looking Plan).
- Rotation Depends on the volume type and the Coordinate System defined, based on the grid section's X and Y size. For example, a volume placed by four points in the Space Management task will rotate the view.
- Scale Set to 1/4" = 1' or 1 = 50.
- North Arrow One symbol included per drawing view. Click and drag the symbol to the Note Area.
- Annotation The plan uses matchline labels to indicate the appropriate coordinate of that segment of the grid boundary along with continuations. The matchlines are lines that outline the boundaries of a grid by following the exterior boundaries of the collection of sectioning elements defining the grid represented in the view.
- **Drawing Volumes** The drawing volume is the gueried 3D volume in the model.

View Styles

The view style specifies the object filters included in the drawing if present in the queried 3D volume. It specifies how objects are displayed, including graphical representation, labeling, and dimensioning. In the **Equipment Plan**, the volume and composed drawing view style definitions are the same.

Title Blocks

The title block generally displays at the bottom of a drawing template. It can include signatures, revision and issue information, and other properties associated with the drawing.

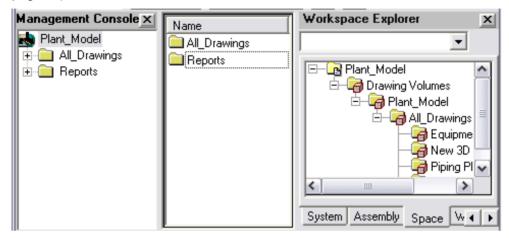


You can add drawing properties to the title block using the **Place Drawing Property Label** command when editing a template.

SECTION 3

Interface Overview

This task includes different windows or views within its interface. You can toggle the display of these windows using commands on the **View** menu. For more information, see *View Menu* (on page 27).



The **Management Console** contains a hierarchy of folders and components that you create. If you right-click an item in the **Management Console**, the available menu commands vary, depending on the item and your permissions. For more information on managing folders and components in the **Management Console**, see *Shortcut Menus* (on page 34). The **Management Console** only displays in the Drawings task.

The **Detail View** shows the children of the selected item in the **Management Console**. You can select multiple components or documents by pressing **Ctrl** or **Shift** while selecting. To specify the columns in the **Detail View**, right-click a column heading and click **More**. The **Detail View** is overlaid by other windows depending on the current operation. For example, when reviewing drawings using the **Open** command, a 2D viewer displays. When you edit report templates, a tabular editor displays. For more information on setting the appearance of the Detail View, see *Detail View (View Menu)* (on page 29).

In plant mode, the **Workspace Explorer** is the tabbed view of systems, assemblies, spaces, and Work Breakdown Structure (WBS) items in the software. For more information about the **Workspace Explorer**, see the *Common User's Guide* available from the **Help > Printable Guides** command in the software. The **Workspace Explorer** does not display in the Drawings task.

NOTE Another window you use while working in this task is the **SmartSketch Drawing Editor** window, which displays as a separate application window. It allows you to edit border templates, drawing templates, and backing sheets for all types of drawings.

See Also

Menus and Toolbars (on page 23)
Icons for Components and Drawings (on page 23)
Permissions Overview (on page 25)

Menus and Toolbars

In this task, the commands available change according to the active window, selected **Management Console** or **Detail View** item, and the specific workflow.

For example, when you edit a drawing template or open a drawing, you control the template with **SmartSketch Drawing Editor**. You use **SmartSketch Drawing Editor** menus and toolbars to edit the open drawing template. When you edit report templates, the report menus and commands are available.

In addition, the shortcut menu that displays when you right-click an item in the **Management Console** or in the **Detail View** differs according to the type of item. For example, some of the commands on the shortcut menu for an Isogen isometric drawing are different from the commands on the shortcut menu for a composed drawing component.

For the root node of the **Management Console** hierarchy and Folder items that have no child items beneath them, the shortcut menu includes **Batch...**, **Copy**, **Paste**, **Delete**, **Rename**, **New...**, **Save Package...**, and **Properties**. If there are any child items beneath the root node or beneath the selected folder, the following commands are added: **Create Drawing(s)**, **Refresh**, **Run Query**, **Update Now**, **Print**, and **Save As**. If you are registered with SmartPlant Foundation, the **Publish** and **Revise** commands are added as well.

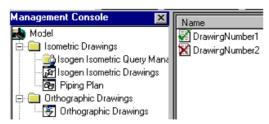
Also, keep in mind that the main menu bar available in the application varies by task. Some commands available in other tasks may not be available in this one.

See Also

Interface Overview (on page 22)
Detail View (View Menu) (on page 29)

Icons for Components and Drawings

The Management Console and Detail View display different icons to show the type and status of packages, components, and documents.



Drawing Type Icons



- Root of the model hierarchy





- Search folder



- Imported folder



- Query manager



- Generic module folder



- Drawing by Rule component (marine mode and material handling mode only)



- Volume Drawing component



- Composed Drawing component



- Orthographic Drawing component



- Isogen Isometric Drawing component



- Spreadsheet Report component



- 3D Model Data component



- MicroStation 3D DGN drawing component

Package of drawing components (in the Add Component dialog box)

- Drawings by Rule package (in the Add Component dialog box) (marine mode and material handling mode only)

🖺 - Drawing document. A status icon is always superimposed over this icon.

Document Status Icons

These icons are superimposed on the document icon and indicate document status.

- Submitted or scheduled for batch processing. These documents are either submitted or scheduled for batch processing.
- Updating or publishing. A document also displays this icon if an error occurred and forced the machine to end the update process before it completed. Right-click the drawing document, and select **View Log** for more information, or update the document again.
- Out-of-date. This document has been altered in SmartSketch Drawing Editor or the drawing properties have been changed. A document is not marked out-of-date if the 3D model has been changed. For example, changing the view style of a drawing view causes the document to be out-of-date, while moving a pump in the 3D model does not affect the drawing status (unless a Refresh is performed).
- Up-to-date. This document is an accurate representation of the 3D model based on the last update performed. If an object in the 3D model that is included in the drawing view has been moved inside the 3D environment, the document is still up-to-date unless **Refresh** is performed. A change must be made to the drawing properties or inside the drawing in order for it to be marked out-of-date without a **Refresh**.
- Error status. An error has occurred during the update process. Right-click the drawing, and select **View Log** for more information.
- No graphic objects in the model associated with this drawing document. For example, the drawing is a piping isometric drawing document created from a Pipeline System that has no piping parts associated with it.
- The drawing document is a version 6.1 legacy Snapshot drawing. You should use the **Tools** > **Convert Legacy Snapshots** command to convert this document to a Composed Drawing for use in the current version of the software. If you do not convert the legacy snapshot drawing, you cannot perform edit operations on the drawing, including update, revise, and publish.

See Also

Components Overview (on page 95)
Interface Overview (on page 22)
Menus and Toolbars (on page 23)
Batch Update Document(s) (on page 300)
Update Now (on page 85)

Permissions Overview

Your site administrator sets permissions and creates permission groups in the Project Management task. These permissions are used in the different tasks in the software to control user access.

You can see your current permission group in the dropdown box in the upper left-hand corner of the window when in the Drawings and Reports task.



The permission group to which an item belongs can affect the actions allowed against that item. For example, the propagation of properties down the hierarchy, from parent to child, is interrupted when a node or document in a read-only permission group is encountered.

The following list shows the actions relating to drawings and reports that are affected by permission groups:

- Accessing shortcut menu commands in the Management Console and Detail View
- Creating items, such as drawings, drawing views, and drawing volumes
- Propagating properties down through the hierarchy
- Deleting items
- Updating items, such as re-extracting drawings

In addition, access to the SharedContent share on the server computer affects actions such as creating and editing view styles and graphic rules.

See Also

Interface Overview (on page 22) Menus and Toolbars (on page 23)

SECTION 4

View Menu

In This Section

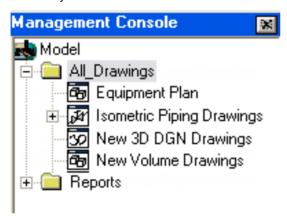
Management Console (View Menu)	27
Detail View (View Menu)	
Workspace Explorer (View Menu)	
Refresh (View Menu)	

Management Console (View Menu)

Toggles the display of the **Management Console**. By default, the **Management Console** is visible when you enter this task.

The **Management Console** contains a hierarchy of folders and components that you create. If you right-click an item in the **Management Console**, the available menu commands vary, depending on the item and your permissions. For more information on managing folders and components in the **Management Console**, see *Shortcut Menus* (on page 34).

NOTE When you switch to a different task and then return to the Drawings and Reports task, the **Management Console** remembers the node you last selected on the console hierarchy.



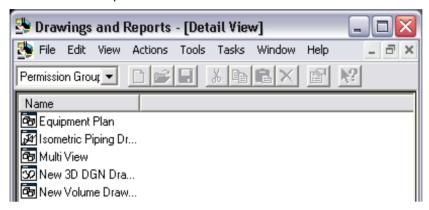
See Also

Interface Overview (on page 22)
Detail View (View Menu) (on page 29)
Workspace Explorer (View Menu) (on page 32)
Refresh (View Menu) (on page 32)

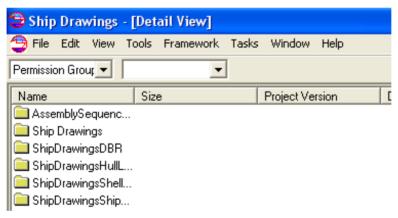
Detail View (View Menu)

Turns the display of the **Detail View** on and off. This command is located on the **View** menu. When checked, the **Detail View** is visible in the application window. When you right-click folders or documents in the **Detail View**, shortcut menus display. The items on the shortcut menu vary depending on the selected item, as shown in the following examples. For more information on the commands, see *Shortcut Menus* (on page 34).

Plant mode example:



Marine mode example:



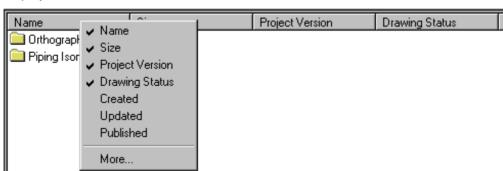
To modify the headings in the **Detail View**, right-click the column-heading area. Select **More** on the shortcut menu to display the **Column Settings** dialog box.

See Also

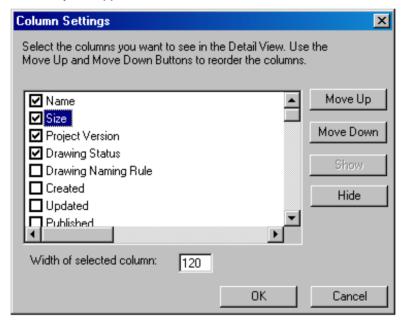
Specify columns in the detail view (on page 30) Column Settings Dialog Box (on page 31)

Specify columns in the detail view

1. Right-click a column heading in the **Detail View**. The shortcut menu shows the currently displayed columns with a checkmark **v**.



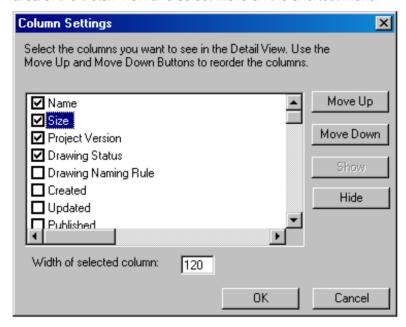
- Add and remove columns automatically by checking and unchecking them on the shortcut menu.
- 3. To modify the appearance and order of the columns, click **More** on the shortcut menu.



- 4. On the **Column Settings** dialog box, select the columns you want to include in the **Detail View**. Clear, or uncheck, the ones you do not want to include. You can also use the **Show** and **Hide** buttons to add and remove columns.
- 5. To change the order of the columns, click **Move Up** and **Move Down** on the **Column Settings** dialog box.
- Specify the width of a column by selecting it and typing an integer in the Width of selected column box. You can also resize columns by dragging the edge of the column in the Detail View.

Column Settings Dialog Box

Specifies the columns you want to see in the **Detail View**. You also can specify the order and width of the columns. You access this dialog box when you right-click in the column heading area of the Detail View and select **More** on the shortcut menu.



Move Up

Moves the selected column up one position. The column displays one position to the left in the **Detail View**.

Move Down

Moves the selected column down one position. The column displays one position to the right in the **Detail View**.

Show

Displays the column in the **Detail View**.

Hide

Hides the column in the Detail View.

▶ NOTE In addition to the **Show** and **Hide** commands, you can use the checkboxes beside the column names to add and remove them from the **Detail View**. Checked indicates that the column displays in the **Detail View**.

Width of selected column

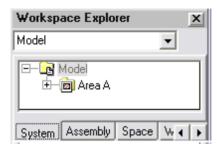
Specifies the width of the column in pixels. You can specify a different column width for each column.

See Also

Interface Overview (on page 22)
Detail View (View Menu) (on page 29)
Management Console (View Menu) (on page 27)

Workspace Explorer (View Menu)

Toggles the display of the **Workspace Explorer** as a viewer only in the Drawings and Reports task. By default, the **Workspace Explorer** is visible when you enter this task. The **Workspace Explorer** displays the contents of the workspace in a classification hierarchy that reflects the various relationships defined for the design objects. The content represents the current objects loaded from the database into the active workspace.



Icons display at the left of the window objects to indicate the type of the object. For example, a file folder icon represents the model, an equipment icon represents equipment, an I-beam icon represents a structural system, and so forth.

See Also

Interface Overview (on page 22)
Detail View (View Menu) (on page 29)
Management Console (View Menu) (on page 27)
Refresh (View Menu) (on page 32)

Refresh (View Menu)

Updates the *loaded* (expanded) content of the **Console** hierarchy, when others are concurrently adding folder, components, or drawings to the hierarchy. The entire hierarchy does not refresh unless you have all the nodes completely expanded. You can also press **F5** to update the content.

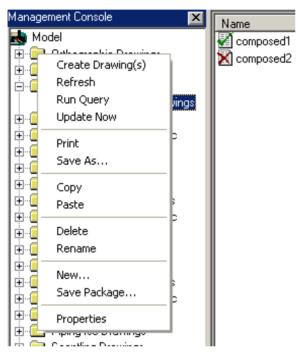
See Also

Shortcut Menus (on page 34)
Interface Overview (on page 22)
Detail View (View Menu) (on page 29)
Management Console (View Menu) (on page 27)

SECTION 5

Shortcut Menus

When you right-click nodes in the **Management Console** and nodes or documents in the **Detail View**, shortcut menus display. The items on the shortcut menu vary depending on the selected item.



Root Shortcut Menu Folder Shortcut Menu Component Shortcut Menu Drawing Document Shortcut Menu

■ NOTES

- By default, the Management Console and Detail View display when you enter the task.
 You can turn the views on or off on the View menu.
- When you switch to a different task and then return to the Drawings and Reports task, the Management Console remembers the node you last selected on the console hierarchy.
- If you are registered with SmartPlant Foundation, the Publish and Revise commands are added as well. You can also set up batch printing and updating for documents using the Batch commands available on the shortcut menus.

See Also

Components Overview (on page 95)

Copy Command

Copies a drawing, component, or folder. After you copy an item, you can paste it at another location in the hierarchy.

The software enforces the following rules regarding pasting:

- If you copy a drawing, select a component to use the Paste command. You cannot paste a
 drawing to a folder or under the drawing book
 - NOTE You can copy only the composed drawings and Drawing by Rule drawings.
- If you copy a composed drawing and paste it on a Composed Drawing component, the Paste Special dialog box is displayed. This dialog box gives you the option to paste the drawing with or without the volume associated with the view. For more information, see Paste Special Dialog Box (on page 43).
 - **NOTE** The **Paste Special** dialog box is displayed for a composed drawing only if it is copied for drawings containing views with associated volumes.
- If you copy an application component, select a folder to use the Paste command. You cannot paste an application component under the drawing book.
- If you copy a folder, select a folder or a drawing book to use the Paste command.
- If you copy a volume or a MicroStation 3D DGN drawings component with a template or a drawing volume associated with it, the **Paste Special** dialog box is displayed if you paste the component in another volume or a MicroStation 3D DGN drawings component. This dialog box gives you the following options to paste: 1) only the template, (2) only the volume, or (3) both the template and the volume. The volume associated with the current volume drawings component being pasted, is copied and associated with the new volume drawings component. For more information, see *Paste Special Dialog Box* (on page 43).
 - **NOTE** A component cannot be pasted under another component.

Copy an item

- 1. Select an item in the Management Console hierarchy or Detail View.
- 2. Right-click the item, and then select **Copy** on the shortcut menu.
- **NOTE** You can paste the copied item at another location in the **Management Console** hierarchy. For more information, see *Paste an item* (on page 42).

See Also

Shortcut Menus (on page 34) Components Overview (on page 95)

Create Drawing(s) Command

Generates the drawings that have not previously been created. This command is available on the right-click menu for various items in the **Management Console** hierarchy.

If you select the top-level of the hierarchy, this command generates all drawings not already created for all components in the hierarchy. For example, if you have Volume Component drawings that have not been created and Isometric drawings that have not been created, both are created if you right-click the top-level hierarchy and select **Create Drawing(s)**.

You can also right-click individual components or folders for which drawings are not yet created and select **Create Drawing(s)** on the shortcut menu to generate the drawings.

After you create drawing documents, you update them to include model object content. You can then open or edit them as needed.

NOTE Marine mode or material handling mode only: If you did not previously select the model contents for a drawings-by-rule component, the **Filter Properties for Asking Filter** dialog box displays, and you must select a filter. The dialog is the same as the **Filter Properties** dialog box, except that only the appropriate tabs for the asking filter are displayed. For more information, see Filter Properties Dialog Box.

See Also

Updating Documents (on page 82) Shortcut Menus (on page 34) Run Query (Shortcut Menu) (on page 111)

Delete Command

Removes an item and its sub-items from the hierarchy and the database. You access this command on the right-click menu for any node or document in the hierarchy. **Undo** is not available for this action. Upon deletion, a confirmation message displays.

When you delete a drawing, its associated template and its component remain unchanged. Any associated physical volumes are deleted.

You can delete many items that are directly or indirectly related to this task:

- Components in the Management Console hierarchy
- Documents
- Drawing volumes
- Drawing views

In some cases, deleting an item causes other items to be deleted. For example, when you delete a drawing volume, associated views are deleted. When you delete a drawing view in a generated drawing, associated volumes and documents are deleted.

In addition, the item status and your permissions can affect whether or not you can delete the item. A drawing set to Approved cannot be deleted. However, you may be able to delete a drawing set to Working.

NOTE You can select multiple components or documents in the **Detail View** and use the **Delete** command from the right-click menu to remove those items.

Delete an item

- 1. Right-click a folder component, document, drawing volume, or drawing view.
- 2. On the shortcut menu, click **Delete**.
- WARNING You cannot undo a delete operation.
- **NOTE** The **Delete** command propagates down the hierarchy. For example, if you delete a volume component, its child components (if any) and all the drawings contained in the components are deleted as well. However, when you delete a single volume drawing, the associated template, volume, and component remain unchanged. You can update the drawing component to re-create the drawing.

See Also

Shortcut Menus (on page 34) Components Overview (on page 95)

Edit Command

Activates the selected drawing for editing. This command is available on the right-click menu for a drawing in the **Detail View**. The drawing opens in **SmartSketch Drawing Editor** with additional commands or toolbars available for editing the drawing.

Objects like drawing views, key plan views, report views, and drawing property labels are placed on the **DwgTemplate** layer when you save the drawing document. You should not place manual markups on the **DwgTemplate** layer. If you use native **SmartSketch Drawing Editor** commands (such as **Place Line** or **Place Dimension**) to add manual markups to the template, put them on the **Default** layer or on a layer that you define (for example, a layer named **AnnotationLayer**). This preserves the changes when you update drawings. For more information on layers, see *Layers* (*SmartSketch Drawing Editor Tools Menu*) (on page 488).



Your access permissions, defined in the Project Management task, affect whether or not you can edit documents.

New Command

Adds a new folder, a search folder, an imported folder, or a component to the **Management Console** hierarchy. Select the model root or a folder and right-click to select **New**. If you select the model root, **New** creates a folder, a search folder, or a folder containing a set of drawing components. If you select a folder, New creates a folder, a search folder, an imported folder, or a drawing component.

Search Folders (on page 272)

Imported Folders (on page 279)

Components Overview (on page 95)

If you have previously created a package, the package is available in the **Add Component** dialog box to add a component to the hierarchy. For more information, see *Save Package Command* (on page 81).

You can configure the shortcut menu for a folder to display more **New** options. For more information, see *Configure New Command On A Folder* in the *Smart 3D Drawings and Reports Reference Data Guide*.

Add Component Dialog Box (on page 40)

What do you want to do?

- Add a folder (on page 39)
- Add a package of components (on page 39)
- Add a component (on page 40)

Add a folder

 Right-click the root model in the Management Console hierarchy, and select New > Folder.

The folder is added to the hierarchy.

2. To rename the folder, right-click the folder, and select **Rename**, or select the folder, and press **F2** on the keyboard. Type a new name.

■ NOTES

- You can place another folder under a folder in the hierarchy.
- You can place a folder, application component, or a package under a folder in the hierarchy.
 For example, you can add an Isogen Isometric Drawings by Query component to a folder.
- You cannot add folders or other components to a search folder component. For more information, see Search Folders (on page 272).

Add a package of components

- 1. Right-click the root model in the **Management Console** hierarchy, and select **New**.
- 2. On the shortcut menu, click **More** to open the **Add Component** dialog box, and select a package for a set of related components. For more information, see *Add Component Dialog Box* (on page 40).
 - A folder and components are added to the hierarchy.
- 3. To rename the folder or a component, right-click and select **Rename** on the shortcut menu, or press **F2** on the keyboard.
- **NOTE** Right-click a component to access the available commands for that component.

Add a component

- 1. In the **Management Console**, create a folder or select an existing folder.
- 2. Right-click the folder, then select **New**.
- 3. On the shortcut menu, select a component such as **Composed Drawings** or **Orthographic Drawings by Query**, or click **More** to open the **Add Component** dialog box and select a component. For more information, see *Add Component Dialog Box* (on page 40).
 - The component is added to the folder.
- 4. To rename the component, right-click the component, and select **Rename**, or select the component, and press **F2** on the keyboard. Type a new name.

■ NOTES

- Right-click a component to access the available commands for that component.
- You cannot add folders or other components to a search folder component. For more information, see Search Folders (on page 272).

Add Component Dialog Box

Lists the available packages and folder types you can use to create folders and drawing components at the selected level in the **Management Console** hierarchy. You access this dialog box when you right-click a folder or the model root, and select **New > More**.

Packages are organized in tabs by drawing type. Select a tab, and then select a package to view its description.

NOTE Many packages are delivered with the software. Your catalog administrator can add new tabs and packages to the dialog box using the **Save Package Command**. For more information, see *Save Package Command* (on page 81).

See Also

Components Overview (on page 95)
Delivered Drawing Types (on page 20)
Icons for Components and Drawings (on page 23)

Open Command

Opens the selected document for viewing within this task. This command is available on the shortcut menu for all document types except 3D Model Data. You also can open the document by double-clicking it.

NOTE Opening a MicroStation 3D DGN document requires that the MicroStation J (V7) or MicroStation J (V8) application be loaded on the workstation.

Open a document

- 1. In the **Detail View**, double-click a document. You can also right-click the document, then select **Open**.
- 2. Close a document by clicking File > Exit.

NOTE To edit the document, right-click the document, the select **Edit**. If the document is a drawing, you can annotate it.

See Also

Open Command (on page 41) Shortcut Menus (on page 34) Components Overview (on page 95)

Paste Command

Inserts the contents that you last copied or cut into the hierarchy or, creates a copy of a drawing volume, depending on the component that you have selected. You must copy or cut an item using the **Copy** or **Cut** command before you can paste it.

The software enforces the following rules regarding pasting:

• If you copy or cut a drawing, select a component to use the **Paste** command.

■ NOTES

- You can copy only the Composed or the Drawing by Rule drawings.
- You can cut only the composed drawings.
- If you paste a composed drawing on a Composed Drawings component, the software displays the Paste Special dialog box. This dialog box gives you the option to paste the drawing with or without the volume associated with the view. For more information, see Paste Special Dialog Box (on page 43).
 - **NOTE** The **Paste Special** dialog is displayed only for the composed drawings if it is copied for the drawings containing views with associated volumes.
- If you copy a folder, select a folder or a drawing book to use the Paste command. You cannot paste a folder to an application component. Application components cannot have a folder below them in the hierarchy.
- If you copy an application component and the Paste command is selected on a folder, the copied application component is pasted under the folder.
- If the application component you copy is a volume drawing or a MicroStation 3D DGN component, and if the component has a template or a drawing volume associated with it, the Paste Special dialog box is displayed. The dialog box gives the option to paste the component with or without the template or the volume of that component.
- If you copy a volume drawings component with a template or a drawing volume associated with it and paste it in another volume drawings component, the software displays the **Paste Special** dialog box. This dialog box gives you the following options to paste: (1) only the template, (2) only the volume, or (3) both the template and the volume. For more information, see *Paste Special Dialog Box* (on page 43).
- If you copy a MicroStation 3D DGN component with a drawing volume associated with it, the software displays the **Paste Special** dialog box if you paste onto another MicroStation 3D DGN component.

Paste Special Dialog Box (on page 43)

Paste an item

Before pasting an item, you must copy or cut the item using the **Copy** or the **Cut** command. For more information, see *Copy an item* (on page 35) and Cut an item.

- 1. Select a location in the Management Console hierarchy or Detail View.
- 2. Right-click the location, and click Paste.

The software pastes the item under the selected location.

Paste Special Dialog Box

Specifies the items to paste if you copied a volume drawing or MicroStation 3D DGN component that has a template or drawing volume defined. The options provided to you depend on the component selected.

If you are pasting a hierarchy containing one or more volume drawings or MicroStation 3D DGN components into a folder, you can select one of the following options:

Copy Nodes(s), Template(s), and Volume(s)

Inserts the new components, including their respective template and drawing volumes, under the selected folder.

Copy Node(s) and Template(s)

Inserts the new components, including their respective templates, under the selected folder.

Copy Node(s) Only

Inserts the new components under the selected folder.

If you are pasting a volume drawings component onto another volume drawings component, or a MicroStation 3D DGN component onto another MicroStation 3D DGN component, you can select one of the following options:

Copy Template(s), and Volume(s)

Copies the template settings and drawing volumes to the selected component.

Copy Template(s) only

Copies only the template settings to the selected component.

Copy Volumes(s) only

Copies only the drawing volumes to the selected component.

If you are pasting a composed drawing on a Composed Drawing snap-in, the following options are available:

Drawing(s) only

Copies only the drawing and its views

Drawing(s) and Associated Volume(s)

Copies the volume associated with the view along with the drawing.

See Also

Paste Command (on page 41)

Print Command

Sends a print request for the selected documents to the default printer. This command is not available until you have created and updated documents.

What do you want to do?

- Print a document (on page 44)
- Print a document as a PDF file (on page 44)

Print a document

- 1. To specify a printer, select **File** > **Select Printer**. For more information on selecting a printer, see *Select Printer (File Menu)* (on page 44).
- Select a folder, application component, or the root node in the Management Console to
 print all of the drawing and report documents beneath the selected level. You can also
 select a single document or multi- select documents in the Detail View. You can select
 multiple documents to print by pressing Ctrl or Shift and then clicking each document in the
 Detail View.
- 3. Right-click and select **Print** on the shortcut menu.

You can also use the **Batch** > **Print** command on the item's shortcut menu to print the item using a scheduled batch job. For more information, see *Batch Print* (on page 298).

Print a document as a PDF file

- 1. To specify a printer, select **File** > **Select Printer**. For more information on selecting a printer, see *Select Printer (File Menu)* (on page 44).
- 2. Select SmartPlant PDF Converter 4xx as the printer.
- Select a folder, application component, or the root node in the Management Console to print all of the drawing and report documents beneath the selected level. You can also select a single document or multi-select documents in the Detail View. You can select multiple documents to print by pressing Ctrl or Shift and then clicking each document in the Detail View.
- 4. Right-click and select **Print** on the shortcut menu.

You can also use the **Batch** > **Print** command on the item's shortcut menu to print the item using a scheduled batch job. For more information, see *Batch Print* (on page 298).

Select Printer (File Menu)

Specifies a printer for documents. The command lists all printers available to your computer.

■ NOTES

- To print a drawing as a PDF, select **SmartPlant PDF Converter 4xx** as the printer.
- **SmartPlant PDF Converter 4xx** requires special settings for 64-bit versions of Windows. For more information, see *Configure SmartPlant PDF Converter for Windows* (on page 45).

See Also

Select Printer Dialog Box (on page 45)

Configure SmartPlant PDF Converter for Windows

Smart 3D installs **SmartPlant PDF Converter 4xx**, a printer driver used to convert drawings and reports to PDF files. Special settings are needed for this driver.

Windows 7

1. Open Start > Control Panel

The Control Panel displays.

- 2. In the Hardware and Sound section, click View devices and Printers.
- 3. Right-click SmartPlant PDF Converter 4xx and select Printer properties.

The SmartPlant PDF Converter 4xx Properties dialog box displays.

- 4. On the Advanced tab:
 - Select Spool print documents so program finishes printing faster.
 - Select Start printing immediately.
 - Clear Enable advanced printing features.
- 5. Click OK.

Select Printer Dialog Box

Specifies a printer for documents.

Name

Specifies a printer name.

Status

Displays the current status of the specified printer.

Type

Displays the type of printer.

Where

Displays the port or location the printer uses.

Setting Properties

The software updates properties from parent nodes to child nodes and drawings in the **Management Console** hierarchy.

For example, you can display the **Properties** dialog box for a folder named **Isometric Drawings**. If you set the **Division Location** property to **Huntsville**, **Alabama**, the software pushes this value to the items contained within the **Isometric Drawings** folder.

You can specify inheritance for each item on its **Properties** dialog box. If you set the override flag for a property, the property is not inherited from the parent. You can provide a new, overriding value for the property. This new value then propagates to other items deeper in the hierarchy.

Properties and Publishing

Several document properties impact publishing the document. Before you can publish documents in the software, you must configure your computer. The configuration includes installing the SmartPlant Client and SmartPlant Schema Component and registering through the SmartPlant Registration Wizard. For more information about the SmartPlant Registration Wizard, see the *Intergraph Smart*TM 3D *Installation Guide*, available from **Help** > **Printable Guides**.

Even if you have registered your model using the SmartPlant Registration Wizard, you must set certain properties to enable the publishing capability. Properties that control publishing are found on the **WBS Tab**. For more information, see *Set properties for publishing documents* (on page 521).

■ NOTES

- The software considers blanks or cleared values as override flags.
- If the drawing document you are looking at in the **Detail View** has a yellow icon (for example:), the drawing document is a version 6.1 legacy Snapshot drawing. You should use the **Tools** > **Convert Legacy Snapshots** command to convert this document to a Composed Drawing for use in the current version of the software. If you do not convert the legacy snapshot drawing, you cannot perform edit operations on the drawing, including update, revise, and publish.

See Also

Edit document properties (on page 47) Properties Command (on page 47)

Properties Command

Views and edits properties for the selected document. The properties of child items are inherited from the parent item unless you set the **Override** column for the properties.

Properties Dialog Box (on page 48)

What do you want to do?

- Edit document properties (on page 47)
- Set surface styles and aspects for 3D model data documents (on page 47)
- Set properties for publishing documents (on page 521)

Edit document properties

- 1. In the **Management Console** or **Detail** view, right-click an item in the hierarchy, and select **Properties** on the shortcut menu.
 - The **Properties** dialog box displays.
- 2. Change the properties as needed. For example, you can set the coordinate system for the drawing on the **Style** tab or view the current approval information on the **Signature** tab.

■ NOTES

- If you do not want an item to acquire a property from its parent, select the **Override** column on the **Properties** dialog box and type a new value. This value propagates to other items deeper in the hierarchy.
- The software treats blank or cleared property values as overrides.

Set surface styles and aspects for 3D model data documents

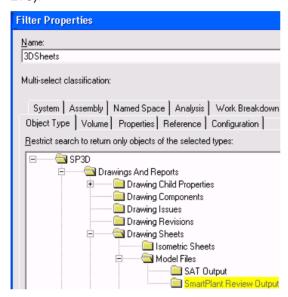
Surface Style Rules and **Aspects** must be set on each 3D Model Data document before updating it using the **Update**, **Update Now**, or **Batch** > **Update** command.

- 1. Select one or more 3D Model Data documents.
- 2. Right-click the selected documents, and select **Properties** on the shortcut menu.
 - The **Properties** dialog box displays.
- 3. Go to the **Surface Styles and Aspects** tab to add surface style rules to the selected rules list. You can also import the surface styles used in the session file. For more information, see *Surface Styles and Aspects Tab (Properties Dialog Box)* (on page 56).
- 4. Click Select Aspects to specify the aspects used within the 3D Model Data documents. If you select no aspects for your documents, the Simple Physical aspect is automatically applied by default. For more information, see Select Aspects Dialog Box (on page 58).
- 5. Click **OK** on the **Properties** dialog box to save the changes to the document(s).

Now you can update the drawing documents using the **Update Now** or **Batch** > **Update** command to incorporate the surface styles and aspects with the 3D Model Data.

Use a Search Folder to Collect 3D Model Data Documents for Property Update

You can create a Search Folder that filters for the 3D Model Data documents so you can edit their surface style rule and aspect settings collectively. When you setup the **Search Folder**, use a filter that looks for the **SmartPlant Review Output** objects. You could additionally specify properties on the output objects to further narrow the search criteria, such as **Data Created**, **Date Last Modified**, or **Signature**. For more information, see *Search Folder Filters* (on page 275).



Properties Dialog Box

Sets options for items in the **Management Console** hierarchy. All items in the **Management Console** have a **Properties** command on their right-click shortcut menus. Using the **Properties** dialog box, you can control how you want properties to propagate through the hierarchy.

You can specify labels for some of the properties on the tabs. Click the browse button at the right of the table cell to display the **Choose Label** dialog box.

General Tab (Properties Dialog Box) (on page 49)

Title Area Tab (Properties Dialog Box) (on page 50)

Signature Area Tab (Properties Dialog Box) (on page 53)

Style Tab (Properties Dialog Box) (on page 55)

Surface Styles and Aspects Tab (Properties Dialog Box) (on page 56)

Custom Tab (Properties Dialog Box) (on page 63)

Notes Tab (Properties Dialog Box) (on page 63)

Issue Tab (Properties Dialog Box) (on page 65)

Revision Tab (Properties Dialog Box) (on page 66)

WBS Tab (Properties Dialog Box) (on page 68)

Configuration Tab (Properties Dialog Box) (on page 69)

■ NOTES

- The WBS tab is available only if you have registered your model using the SmartPlant Registration Wizard.
- If your model has not been registered using the SmartPlant Registration Wizard, the Issue tab is always available. If your model has been registered, the Issue tab is available only if you have issued documents to a contract and the information is read-only. For more information, see Issue request documents (on page 529).

See Also

Setting Properties (on page 46)

General Tab (Properties Dialog Box)

Shows general properties for a drawing item.

Name

Displays the name of the property.

Value

Sets the current value of the property.

Behavior

Specifies whether to inherit or override a property in the hierarchy of items in the **Console**. If the property is **Inherited**, the setting comes from items higher in the hierarchy. If the property is not read-only, you can provide a new, overriding value for the property by changing the behavior setting to **Override**. This new value then propagates to other items deeper in the hierarchy. If you set the behavior to **Force Override**, the property setting is forced to items deeper in the hierarchy, even if they are set to **Inherit**.

Properties

Published

Indicates whether the drawing has been published.

Size

Displays the size of the file in KB (kilobytes).

See Also

Properties Dialog Box (on page 48)

Title Area Tab (Properties Dialog Box)

Sets options for the title area of drawings.

If you access this tab after selecting multiple drawings, these fields will appear empty, regardless of what information was defined for any of the drawings individually. Any information you add to this tab will overwrite the previously defined information in corresponding rows in the selected drawings.

Name

Displays the name of the property.

Value

Sets the current value of the property.

Behavior

Specifies whether to inherit or override a property in the hierarchy of items in the **Console**. If the property is **Inherited**, the setting comes from items higher in the hierarchy. If the property is not read-only, you can provide a new, overriding value for the property by changing the behavior setting to **Override**. This new value then propagates to other items deeper in the hierarchy. If you set the behavior to **Force Override**, the property setting is forced to items deeper in the hierarchy, even if they are set to **Inherit**.

Properties

Border

Specifies the border attribute that stores the name of border for the title block. This attribute also stores the dimensions of the border.

Charge Number

Defines the charge number for the drawing.

Charge Title

Describes the **Charge Number** box. The charge title text is placed to the left of the charge number.

Company Name

Specifies the name of the company for which the project is designed.

Desc1

Describes the drawing. This description is the first of four lines of text used to describe the drawing.

Desc2

Describes the drawing. This description is the second of four lines of text used to describe the drawing.

Desc3

Describes the drawing. This description is the third of four lines of text used to describe the drawing.

Desc4

Describes the drawing. This description is the fourth of four lines of text used to describe the drawing.

Division Location

Specifies the location of the division for which the project is designed.

Division Name

Specifies the name of the division for which this project is designed.

Drawing Naming Rule

Specifies a default or custom naming rule to the drawing. The default name rules provided include:

Default Drawing Name Rule - This is the default rule for composed drawing types. Creates a name based on the parent component name, Global Workshare location ID, and an index number. The naming rule inserts a "-" between each name. If there are no parent objects, then only the child object name is used. For example, the first drawing created under the ComposedDrawings001 component at workshare site 1 is called *ComposedDrawings001-1-0001*.

Default By Query Name Rule - This is the default rule for all Drawings by Query drawing types. Creates a name based on the parent object name (each system and sub-system above the child object will be included in the name) and child object name. The naming rule inserts a "-" between each name. If there are no parent objects, then only the child object name is used. For example, the first composed drawing created at workshare site 1 under the ComposedDrawings001 component would be named *ComposedDrawings001-0001*.

Default Report Name Rule - This is the default rule for all Spreadsheet Reports. Creates a name based on the object name, Global Workshare location ID, and an index number. For example, the first drawing created at workshare site 1 is called *ComposedDrawings001-1-0001*.

Volume Name Rule - This is the default rule for volume drawing types. Creates a name based on the volume name, Global Workshare location ID, and an index number. For example, the first drawing created using the volume Volume001 in workshare site 1 is called *Volume001-1-0001*. If a volume is not specified, the drawing name will be named *Unspecified*.

NOTE Customized naming rules appear in the list if you bulkload against the **CDrawingSheet** class, which is the class for the drawing object. The rules are defined on the **NamingRules** sheet in the *GenericNamingRules.xls* workbook. For more information, see the *Reference Data Guide*.

Drawing Number

Displays the unique identifier for the drawing.

Drawing Size

Defines a standard note value for the border size.

Drawing Type

Defines the three-letter code to identify the type of drawing. For example, the type can be DGN.

Job Number

Defines the unique identifier assigned to a capital project or job.

Note Line

Specifies text for a miscellaneous note line.

Plant Name

Specifies the name of the plant or project for which the drawing is designed.

Project Version

Defines the number and letter sequence that identifies a particular generation of a document that was created since the last approved revision.

Revision Numbers

Defines the number of the current revision for this drawing.

Sheet

Defines the number of the page and the total number of pages that are associated with this one. For example, the value might be **3 of 5**.

Site Location

Specifies the site location for which the drawing is designed.

Site Name

Specifies the name of the site where the plant is being constructed.

Title1

Specifies text for the first miscellaneous title. This title is usually a description of the area shown on the drawing.

Title2

Specifies text for the second miscellaneous title. This title is usually a description of the type of drawing.

Title3

Specifies text for the third miscellaneous title.

See Also

Properties Dialog Box (on page 48)

Signature Area Tab (Properties Dialog Box)

Sets options for the signature area of drawings.

If you access this tab after selecting multiple drawings, these fields will appear empty, regardless of the information defined for any of the drawings individually. Any information you add to this tab overrides the previously defined information in corresponding rows in the selected drawings.

Name

Displays the name of the property.

Value

Sets the current value of the property.

Behavior

Specifies whether to inherit or override a property in the hierarchy of items in the **Console**. If the property is **Inherited**, the setting comes from items higher in the hierarchy. If the property is not read-only, you can provide a new, overriding value for the property by changing the behavior setting to **Override**. This new value then propagates to other items deeper in the hierarchy. If you set the behavior to **Force Override**, the property setting is forced to items deeper in the hierarchy, even if they are set to **Inherit**.

Properties

Approval Date

Specifies the date the drawing was approved.

Approved By

Specifies the name of the person responsible for approving the drawing.

Checked By

Specifies the name of the person responsible for checking the drawing.

Checked Date

Specifies the date the drawing was checked.

Designed By

Specifies the name of the person who specified or designed the information on the drawing.

Designed Date

Specifies the date the drawing was designed.

Drawing Status

Defines the status code for the drawing.

Drawn By

Specifies the name of the person who drew the drawing, or created it.

Drawn Date

Specifies the date the drawing was drawn or created.

Extra Sign By1

Specifies the name of an extra person who is signing the drawing.

Extra Sign By2

Specifies the name of an extra person who is signing the drawing.

Extra Sign Date1

Specifies the date the drawing was signed by the person whose name displays in this box.

Extra Sign Date2

Specifies the date the drawing was signed by the person whose name displays in this box.

Extra Sign Title1

Defines the title of the person whose name displays in this box.

Extra Sign Title2

Defines the title of the person whose name displays in this box.

Mfg Rep Date

Specifies the date that the manufacturing representative initials the drawing.

Mfg Rep Name

Specifies the name of the manufacturing representative who signed the drawing.

Plant Number

Defines the plant number.

Proj Engineer Date

Specifies the date the project engineer initials the drawing.

Proj Engineer Name

Specifies the name of the person who is the project engineer for the project using the drawing.

Spec By

Specifies the name of the person who specified or designed the information on this drawing.

Spec Date

Specifies the date this drawing was specified or designed.

See Also

Properties Dialog Box (on page 48)

Style Tab (Properties Dialog Box)

Sets options for the style of drawings and reports.

If you access this tab after selecting multiple drawings, these fields will appear empty, regardless of the information defined for any of the drawings individually. Any information you add to this tab overrides the previously defined information in corresponding rows in the selected drawings.

Name

Displays the name of the property.

Value

Sets the current value of the property.

Behavior

Specifies whether to inherit or override a property in the hierarchy of items in the **Console**. If the property is **Inherited**, the setting comes from items higher in the hierarchy. If the property is not read-only, you can provide a new, overriding value for the property by changing the behavior setting to **Override**. This new value then propagates to other items deeper in the hierarchy. If you set the behavior to **Force Override**, the property setting is forced to items deeper in the hierarchy, even if they are set to **Inherit**.

Properties

Coordinate System

Specifies the global or an active coordinate system. Choose a coordinate system from the list, or click **More...** to choose another coordinate system with the **Select System** dialog box.

If you want to output large coordinates on your drawings, define a coordinate system using large negative coordinates. For example, if you want coordinates of **400,000 ft** output on drawings, define a coordinate system origin of **-400,000 ft** and place your model elements close to **global 0**. Select the new coordinate system in the **Coordinate System** field on the **Style** tab. For more information on defining coordinate systems, see the *Grids User's Guide* available from **Help > Printable Guides**.

■ NOTES

- The Coordinate System property is not used by Composed Drawings. The coordinate system settings are driven by the properties for the drawing views in a composed drawing. For more information on the coordinate system properties for a drawing view, see Drawing View Properties Dialog Box (Place View Command) in the SmartSketch Drawing Editor Help.
- If you are accessing the **Properties Style** tab for a 3D Model Data component or document with the intention of saving it as a SmartPlant Review file, make sure this property is set appropriately so that the **Plant Monument Coordinate Offset** is passed correctly to SPR when creating the VUE file. This is because SPR shows the objects from the VUE file using global coordinates. The offset value allows you to see the original coordinates relative to the new SPR coordinate system. For more information on 3D Model Data components, see 3D Model Data. For information on saving to SPR, see *Save as SmartPlant Review File* (on page 508).

Volume Naming Rule

Specifies the naming rule applied to the content of the drawing.

Change Management

Enables and disables **Change Management** for Isogen isometric drawings. You can override the **Drawing.Content.ChangeManagementEnabled** option in the isometric drawing style with this property. You can set the property to **Enabled**, **Disabled**, or **Undefined**. This property is only available when you are viewing properties for an Isometric piping isometric drawing. For more information on Change Management, see *Change Management in Piping Isometric Drawings* (on page 222).

Baseline Date

Identifies a date in time when a *snapshot* of the drawing document was taken. It is a way of identifying when objects have changed.

WBS Project

Specifies the Work Breakdown Structure (WBS) project style to be used with the drawing. This property serves as the answer to an asking filter when specified in a **View Style**. For more information, see Create a Drawing Using WBS Objects.

See Also

Properties Dialog Box (on page 48)

Surface Styles and Aspects Tab (Properties Dialog Box)

Sets options for the surface styles and aspects used in 3D Model Data. This tab is only available when you are looking at the properties for a 3D Model Data document.

You can perform the following operations on this tab:

- Select an existing surface style rule from the library and add it to the workspace.
- Modify an existing surface style rule in the library and add it to the workspace.
- Create a new surface style rule and add it to the library and the workspace.
- Delete a surface style rule from the library or the workspace.
- Rearrange the style rules in the workspace box of the Surface Style Rules dialog box by using the Move Up and Move Down commands.
- Import surface styles from the session file.
- Set the aspects for the 3D Model Data.

Surface style rules

Style rule library

Lists all the current surface style rules in the Site database.

Selected rules

Lists all the names for the surface style rules currently assigned to the workspace.

Add

Adds the selected surface style rule to the workspace.

Remove

Removes a selected surface style rule from the workspace. To remove a surface style from the workspace, select the style in the **Workspace** list and click **Remove**.

Move Up

Moves the selected style rule up one step in the Workspace list.

Move Down

Moves the selected style rule down one step in the **Workspace** list.

New

Activates the **Surface Style Rule Properties** dialog box on which you can create a new surface style rule and add it to the database. This button is available only if you have write permission to the surface style rules.

Modify

Activates the **Surface Style Rule Properties** dialog box to modify an existing surface style rule and add it to the database.

NOTE For more information on creating new or modifying existing surface style rules, see *Surface Style Rule Properties Dialog Box* (on page 60).

Copy

Creates a copy of the selected rule on the Clipboard. You use **Copy** to create a copy of a surface style rule in the Model database so you can modify the rule rather than create a new one.

NOTE If you try to copy a style rule associated with a deleted filter, the style cannot be copied. A message box displays.

Paste

Pastes the copied rule from the Clipboard so it can be modified.

Delete

Removes the selected Surface Style Rule from the database.

Apply

Applies changes in surface style rules to the workspace.

Properties Double-clicking a surface style rule also activates the **Surface Style Rule Properties** dialog box on which you can create or modify a surface style rule if you have permission.

Import from Session

Imports session surface style rules for the selected 3D Model Data component.

Aspects

Select Aspects

Opens the **Select Aspects** dialog box so you can specify the aspects to use for the 3D Model Data component. For more information, see *Select Aspects Dialog Box* (on page 58).

NOTE When publishing 3D Model Data documents, the **Simple Physical** aspect is used by default if no other aspects are specified.

See Also

Properties Dialog Box (on page 48)

Select Aspects Dialog Box

Specifies the aspect associated with the 3D Model Data document(s). An aspect is a geometric area or space related to an object. The aspect represents information about the object, such as its physical shape or the space required around the object. Aspects as associated with parameter for an object, representing additional information needed for placement. Aspects can represent clearances for safety or maintenance, additional space required during operation, or simple and details representations of the object. You define aspects when you model a part class for the reference data.

This dialog box displays when you click the **Select Aspects** button on the **Surface Styles and Aspects** tab of the **Properties** dialog box for the 3D Model Data document(s). **Selected aspects**

Shows a checkbox list of all aspects defined by the model reference data. You can check multiple aspects. By default, the **Simply Physical** aspect is selected.

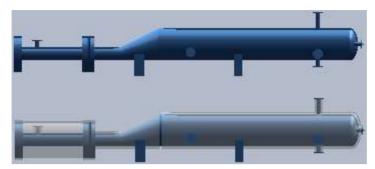
Simple physical – Includes primitive shapes. This aspect creates a less cluttered view
of the object, showing only the body of equipment or a simplified cross-section for
structure, as shown below.



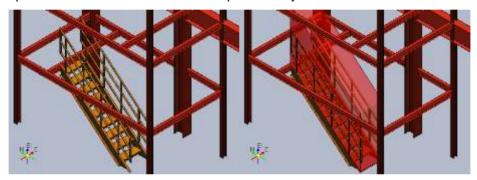
Detailed physical - Provides a more detailed view of an object. This aspect shows all
the graphical details associated with the equipment or structure. For example, certain
types of equipment may include legs and lugs. For marine structure, this aspect uses all
geometry in the cross-section, as shown below.



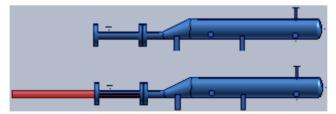
 Insulation - Shows an area around a piece of equipment indicating the presence of insulation. This aspect is also used to display structural fireproofing insulation. For example, a 4-inch pipe with insulation might look like an 8-inch pipe when this aspect is used.



 Operation - Includes the area or space around the object required for operation of the object. This space shows in the model but not in drawings. For example, this aspect leaves enough space around a motor for a person to operate the motor or the overhead space needed for someone to walk up a stairway.

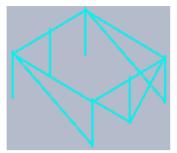


• Maintenance - Includes the area or space around the object required to perform maintenance on the object. This space may appear in the model but not in drawings. For example, this aspect leaves enough space around a motor to perform maintenance on a motor, including space to remove the motor.



- Reference Geometry Allows you to construct or add graphical objects that do not participate in interference checking. For example, a reference geometry object could be a spherical control or the obstruction volume for a door. For marine structure, this aspect allows you to control the display of landing curves for design seams, profile systems, reference curves, and knuckle curves.
- **Centerline** Displays objects as a single line representation. For example, this aspect is useful for when you want to display handrails or structural members as a single-line on

drawings. For structure, the centerline is determined from the cardinal point used to place the member. That cardinal point is not always the center of the object.



- Molded Forms Displays plate, profile, and beam systems for marine structure.
- Equipment Hole Displays holes for marine structure created in the Hole Management task.
- * Matches all cross-sections.

For more information on defining aspects for your model reference data, see the *Common User's Guide*.

See Also

Surface Styles and Aspects Tab (Properties Dialog Box) (on page 56) Box) (on page 60) Configuration Tab (Surface Style Rule Properties Dialog Box) (on page

Surface Style Rule Properties Tab (Surface Style Rule Properties Dialog Box)

Creates or modifies a surface style rule. Surface style rules are based on filters. When you create new rules or modify rules, you specify a filter on which to base the rule. For more information on filters, see the *Common User's Guide*.

Rule name

Specifies the name of the surface style rule.

Filter

Identifies the filter used within the style rule. The filters available are the ones defined for the current database. The list in the dropdown includes the last ten filters selected. Selecting **Create New Filter** in the dropdown list displays the **New Filter Properties** dialog box, which allows you to define a new filter for the style rule. Selecting **More** in the list displays the **Select Filter** dialog box. The **Properties** button for this field displays the **Property** dialog box for the selected filter. For more information on defining a new filter or reviewing properties, see the *Common User's Guide*.

• TIP We recommend that you use simple, asking, and compound filters with style rules. Using SQL filters can result in significant performance degradation and should be avoided whenever possible. Unlike the other types of filters, the SQL server is performed directly on the database. For each object passed to the SQL filter, the software checks to see if any of the objects were returned by the query. However, modification of the object may change whether or not the object passes the SQL filter. For example, a pipeline might pass the SQL filter before it is assigned to a different system. After the system assignment changes, a different style rule is applied. Therefore, some SQL filters may result in decreased efficiency in assessing the project data model.

Style applied

Specifies the surface style to be used for the objects identified by the selected filter. The list in the dropdown includes all surface styles available for the current database. The **Properties** button displays the **Surface Style Rule Properties** dialog box so you can edit the style as needed.

Select all aspects to which the style will be applied

Shows a checkbox list of all aspects defined by the model reference data. You can check multiple aspects. By default, all aspects are selected.

■ NOTES

- An aspect is a geometric area or space related to an object. The aspect represents information about the object, such as its physical shape or the space required around the object. Aspects are associated parameters for an object, representing additional information needed for placement. Aspects can represent clearances for safety or maintenance, additional space required during operation, or simple and detailed representations of the object. You define aspects when you model a part class for the reference data.
- The **Simple Physical** aspect includes primitive shapes. The space can be a field junction box displayed in both the model and in drawings. When you publish 3D Model Data documents, this is the default aspect used if no other aspects are selected for the document properties.
- The Detailed Physical aspect provides a more detailed view of equipment in the model. For example, certain types of equipment may include legs and lugs. As opposed to the Simple Physical aspect, which only shows the body of the equipment, the Detailed Physical aspect shows all the graphical details associated with the equipment.
- The Insulation aspect shows an area around a piece of equipment indicating insulation is present. For example, a 4-inch pipe with insulation might look like an 8-inch pipe when the Insulation aspect is selected.

- The Operation aspect includes the area or space around the object required for operation of the object. This space shows in the model but not in drawings. The Operation aspect leaves enough space around a motor for a person to operate the motor.
- The Maintenance aspect includes the area or space around the object required to perform maintenance on the object. This space may appear in the model but not in drawings. The Maintenance aspect leaves enough space around a motor to perform maintenance on the motor, including space to remove the motor, if necessary.
- The Reference Geometry aspect allows you to construct or add graphical objects that do not participate in interference checking. For example, a reference geometry object could be the obstruction volume for a door on a field junction box. Another example is a spherical control point.

See Also

Surface Style Rule Properties Dialog Box (on page 60)

Configuration Tab (Surface Style Rule Properties Dialog Box)

Displays the creation, modification, and status information of an object.

Model

Displays the name of the model. You cannot change this value.

Permission Group

Specifies the permission group to which the object belongs. You can select another permission group, if needed. Permission groups are created in the Project Management task.

Transfer

Re-assigns ownership of the selected model objects from their current permission group to another satellite or host permission group. This button is only available if the active model/project is replicated in a workshare configuration. The button is not available if all of the objects in the select set already belong to another location and are non-transferable. For more information, see *Transfer Ownership Dialog Box* in the *Structural Detailing User's Guide*.

NOTE The **Transfer** option does not apply to the Surface Style Rules.

Status

Specifies the current status of the selected object or filter. Depending on your access level, you may not be able to change the status of the object.

Created

Displays the date and time that the object was created.

Created by

Displays the name of the person who created the object.

Modified

Displays the date and time when the object was modified.

Modified by

Displays the name of the person who modified the object.

Custom Tab (Properties Dialog Box)

Sets options for drawing properties.

If you access this tab after selecting multiple drawings, these fields appear empty, regardless of the information defined for any of the drawings individually. Any information you add to this tab overrides the previously defined information in corresponding rows in all selected drawings.

Name

Displays the name of the property.

Value

Sets the current value of the property.

Behavior

Specifies whether to inherit or override a property in the hierarchy of items in the **Console**. If the property is **Inherited**, the setting comes from items higher in the hierarchy. If the property is not read-only, you can provide a new, overriding value for the property by changing the behavior setting to **Override**. This new value then propagates to other items deeper in the hierarchy. If you set the behavior to **Force Override**, the property setting is forced to items deeper in the hierarchy, even if they are set to **Inherit**.

See Also

Properties Dialog Box (on page 48)

Notes Tab (Properties Dialog Box)

Sets notes for the item.

If you access this tab after selecting multiple drawings, these fields appear empty, regardless of the information defined for any of the drawings individually. Any information you add to this tab overrides the previously defined information in corresponding rows in the selected drawings.

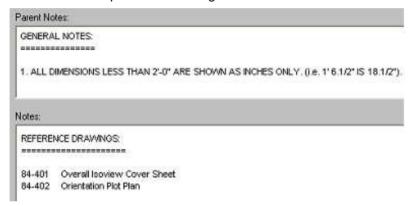
Parent Notes

Concatenates the notes from any parents of the currently selected item and displays the notes. This box is read-only.

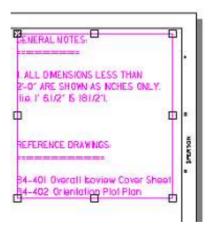
Notes

Specifies notes for the currently selected item.

The following graphics demonstrate how parent notes and notes work. The first graphic shows how notes can be added at different levels of the hierarchy. The **Notes** tab shows Parent Notes from a higher-level folder or component. The Notes section shows additional information for a particular drawing document.



When the document is updated and displayed, the Note Region of the template contains the specified information.



See Also

Properties Dialog Box (on page 48)

Issue Tab (Properties Dialog Box)

Sets options for internal issues.

Issue Number

Shows the drawing issue number. Type the required issue number for the drawing. When you have selected multiple drawings or a node in the **Management Console**, this field is not available to edit, as an issue number is created for each individual drawing, depending on its current revision history.

Description

Describes briefly the scope of the issue.

Issue Date

Shows the date issued.

Issue Reason

Shows the reason the document was issued. Select a value from **Bid**, **Fabrication**, **Construction**, **Reference**, and your customized values, if any are defined.

NOTE The values for **Issue Reason** can be customized by editing and bulkloading the **DrawingIssueReason_Codelist.xIs** workbook delivered in the [Product Folder]\CatalogData\Bulkload\AdditionalDataFiles folder. For more information, see Bulkload Files in the Drawings and Reports Reference Data Guide.

Job Spec

Identifies the job specification for the issued document.

Revision Number

Defines the revision number for this issue of the drawing.

Unregistered

If you access **Properties** on a single document when your model has not been registered using the SmartPlant Registration Wizard, the **Issue** tab displays previous entries. A new row is available to make a new entry. You can edit each field using alphanumeric and special characters. You cannot delete a row after it has been added.

If you access **Properties** on a folder when your model has not been registered using the SmartPlant Registration Wizard, the **Issue** tab has a single blank row for a new entry. With the exception of the **Issue Number** field, you can edit all the fields. Their values are propagated to the documents within the folder.

Registered

If your model is registered using the SmartPlant Registration Wizard and you have issued requests for the document, the **Issue** tab is read-only. The Issue information is retrieved for informational purposes only.

■ NOTES

- You can create only one issue per instance of the **Properties** dialog box. To create another issue, close the dialog box and open it again.
- For information on issuing requests for contracts when working in an integrated environment, see *Issue request documents* (on page 529).

See Also

Properties Dialog Box (on page 48)

Revision Tab (Properties Dialog Box)

Displays and sets properties for revisions. The **Revision** tab is always read-write (subject to user permissions).

Action

Lists the available actions when accessing Properties on a single component, multiple components, or multiple documents.

Append Record - Creates a new revision record for each document under the selected components or in the document set. You can type a value for the next revision mark or let the software automatically increment it for you.

Edit Last Record - Edits the last revision for each document under the selected components or in the document set. Only the edited revision fields overwrite the corresponding fields on the last revision record. To clear a populated revision field, type a single space character, and no other characters, in the edited field.

NOTE The **Append Record** and **Edit Last Record** options are not available for a model registered with SmartPlant Foundation or when revising a single document.

Revision Mark

Specifies the current revision. For single documents, double-click the **New Record** cell to automatically increment to the next revision mark number. To manually type a value for the next revision mark, click the **New Record** cell and type the value. This only applies when the model has not been registered with SmartPlant Foundation. If this cell is not edited, then the revision mark number automatically increments to the next available number in each writeable document associated to the selected set.

Revision Minor Number

Specifies the minor revision number for the revision.

Description

Describes the scope of the revision.

Revised By

Identifies the person who made the revision.

Revision Date

Specifies the date of the revision.

Check

Identifies the person who checked the revision.

Check Date

Specifies the date the revision was checked.

Approved By

Identifies the person who approved the revision.

Approval Date

Specifies the date the revision was approved.

The appearance and behavior of the contents of this tab differ depending on whether properties are accessed on a single document or accessed on a single component, multiple components, or multiple documents. The contents of this tab also depend on whether the model is registered to SmartPlant Foundation.

Unregistered

If you access **Properties** on a single document and your model has not been registered to SmartPlant Foundation, the **Revision** tab displays previous entries made. A new row is available to make a new entry. You can edit each field using alphanumeric and special characters.

If you access **Properties** on a single component, multiple components, or multiple documents and your model has not been registered to SmartPlant Foundation, the **Revision** tab has a single blank row for a new or edited entry. All fields are editable. Their values are propagated to the writeable documents that are associated with the selected set.

Registered

If your model has been registered to SmartPlant Foundation, use the **Revise** command to create revision numbers. This command reserves a revision number by adding it to the document Revision properties. The revision number is added in the form of a blank row on the **Revision** tab of the **Properties** dialog box.

After reserving the revision number, right-click the document and select **Properties**. Go to the **Revision** tab and edit the **Revision** fields. All fields except for **Revision Mark** and **Revision Minor Number** are editable. For more information, see *Revising* (on page 513).

■ NOTES

- You can create more than one revision per instance of the Properties dialog box by selecting Apply after adding a record.
- You can delete one or more revision records by highlighting the revision rows and pressing Delete. You must select OK or Apply to make the deletion permanent. The rows selected for deletion must be adjacent and must include the last revision record.

See Also

Properties Dialog Box (on page 48)

WBS Tab (Properties Dialog Box)

Sets options for the Work Breakdown Structure (WBS) of drawings and reports. This tab is available only when your model has been registered using the SmartPlant Registration Wizard.

If you access this tab after selecting multiple drawings, these fields appear empty, regardless of the information defined for any of the drawings individually. Any information you add to this tab overrides the previously defined information in corresponding rows in the selected drawings.

Name

Displays the name of the property.

Value

Sets the current value of the property.

Behavior

Specifies whether to inherit or override a property in the hierarchy of items in the **Console**. If the property is **Inherited**, the setting comes from items higher in the hierarchy. If the property is not read-only, you can provide a new, overriding value for the property by changing the behavior setting to **Override**. This new value then propagates to other items deeper in the hierarchy. If you set the behavior to **Force Override**, the property setting is forced to items deeper in the hierarchy, even if they are set to **Inherit**.

Properties

Project Name

Displays the project to which the item belongs. In SmartPlant Foundation, a project is the scope of work approved for capital expenditure (that is, a job).

Document Type

Specifies the type of document, such as Civil Plan.

Document Style

Specifies the style of document, such as Ortho for orthographic drawing.

Discipline

Specifies the discipline for the document. If this is a 3D Model Data document, set the property to **SmartPlant Review Document**. If it is a drawing or report document, set the discipline to match the type of document.

Allow Publish

Sets the document as a publishable document.

Working with the Integrated Environment

You can only publish documents after the appropriate properties are set on the **WBS** tab. The **WBS** tab is not available if your login is not authenticated as valid for SmartPlant Foundation. The properties that must be defined for publishing are: **Document Type**, **Document Style**, **Discipline**, and **Allow Publish**. For more information, see *Set properties for publishing documents* (on page 521).

Configuration Tab (Properties Dialog Box)

Displays the creation, modification, and status information for an item.

Model

Displays the name of the Model. This value is read-only.

Permission group

Specifies the permission group to which the item belongs. You can select another permission group if needed.

If you access this tab after selecting multiple drawings, this field will be empty, regardless of what information was defined for any of the drawings individually. Any selection you in this field will overwrite the previously defined information in the selected drawings.

Transfer

Reassigns ownership of the selected model objects from their current permission group to another satellite or host permission group. This option is only available if the active model or project is replicated in a workshare configuration. The option is not available if all of the objects in the select set already belong to another location and are non-transferable. For more information, see *Transfer Ownership Dialog Box* in the *Common User's Guide*.

■ NOTE The Transfer option does not apply to the filters and surface style rules.

Approval State

Shows the approval state of the selected item. This value is read-only.

Status

Specifies the current status of the selected **Console** hierarchy item or items or selected documents in the **Detail View**. Depending on your access level, you may not be able to change the status of the selected items.

Date Created

Displays the date and time the item was created.

Created by

Displays the name of the person who created the item.

Date Last Modified

Displays the date and time the item was modified last.

Last Modified by

Displays the name of the person who modified the item last.

See Also

Properties Dialog Box (on page 48)

Transfer Ownership Dialog Box

Allows you to specify a new location and permission group for the selected model objects.

Current location

Displays the name of the location with which the current permission group is associated. All of the objects in the select set must belong to the same location.

Current permission group

Displays the name of the permission group with which the selected objects are currently associated. If all of the objects in the select set do not belong to the same permission group, this box appears blank.

New location

Specifies the name of the location to which you want to assign the objects. In a global workshare configuration, this box lists all the locations in which you have write access to one or more permission groups. The selection in this box filters the entries in the **New permission group** box.

New permission group

Specifies the new permission group to which to assign the selected objects. If you specify a value in the **New location** box, this list displays all permission groups to which you have write access in the selected location. If you do not specify a value in the **New location** box, this list includes all permission groups to which you have write access in all locations except the current location. This box is blank if you do not have write access to any permission groups at any locations other than the current one.

NOTE We strongly recommend that administrators follow naming convention rules that include the location as a prefix in the permission group name.

Choose Label Dialog Box

Specifies a label for a document property. This dialog box displays the labels available on the application server in the [Reference Data Folder]\SharedContent\Labels\Base Templates folders.

See Also

Setting Properties (on page 46)

Rename Command

Allows you to type a new name for a selected item in the hierarchy. The shortcut key for this command is **F2**.

NOTE You cannot have duplicate names at the same level in the tree, but the names are case-sensitive. For example, you can have two items be named 'ItemName' and 'itemname' at the same level in the tree.

Rename an item

- 1. Right-click an item in the **Management Console** hierarchy or **Detail View**, and select **Rename** on the shortcut menu. Alternatively, press **F2** on the keyboard.
- 2. Type a new name for the item.

See Also

Rename Command (on page 71) Shortcut Menus (on page 34) Components Overview (on page 95)

Save As Command

Saves drawings and reports as specified file types to an external location, such as a share on another server. This command is not available until you generate drawings for at least one of the structures in the hierarchy. This command saves only the structures that contain drawings or reports. You can save multiple file types based on the types of documents available. You can specify the target file type for each drawing type you want to save.

To save the hierarchy as a package, right-click the folder and select **Save Package** on the right-click menu. For more information, see *Save Package Command* (on page 81).

Save As Dialog Box (on page 78)

What do you want to do?

- Save to a file (on page 73)
- Retrieve piping component file data (PCF) (on page 74)
- Save as SmartPlant Foundation (*.xml) (on page 76)

Save to a file

Prior to following this procedure, you must have generated drawings already for at least one of the items in the **Console** hierarchy. The **Save As** command is not available if drawings have not been generated.

- Select a folder, component, or document. You can select multiple documents by holding Ctrl or Shift and clicking each item.
- 2. Right-click your selection, then select **Save As** on the shortcut menu.
 - The Save As Dialog Box (on page 78) displays.
- 3. Specify the **Output Folder Rule** to be used. You can save the item as it displays in the hierarchy, with its parent folder appended or with the entire model hierarchy appended.
- 4. Specify the **File Already Exists Action** to be used. This determines how you save the file if it has the same name of an existing file. Select **Overwrite** to replace the existing file, or select **Save As filename (n)** to save the file separately.
- 5. Specify the **Output Folder** location. Click **Browse** to display a dialog box to locate the appropriate folder location.
- 6. Check the boxes for the **Component Types** you want to save. You can select multiple component types. For more information, see *Save As Dialog Box* (on page 78).
- 7. In the **Target File Type** lists, specify the file types you want to save. You can specify a file type for each component type selected. For example, you could use the Iso_Stress style to create a Piping Component File (PCF) file, then when you perform a **Save As** on the

document, check the **Isometric Drawings** component type and specify the **Target File Type** as **PCF File**.

■ NOTES

- For Isogen isometric drawings, the name of the drawing document becomes the prefix for all of the files. For example, if the drawing document name is *My_Pipeline*, the saved file names become: *My_Pipeline.sha*, *My_Pipeline.pcf*, and so on.
- For Isogen isometric drawings, a file is created for each sheet in the drawing with [drawing name]_[sheet name] as the filename. For example, if the drawing My_Pipeline contains Sheet1 and Sheet2, two files will be saved with the names My_Pipeline_Sheet1 and My_Pipeline_Sheet2.
- If you configured a **StartProgram**, then the extracted PCF file will be stored with the relation *<Filename>-extracted-pcf*. For example, if the final PCF file name is *My_Pipeline*, then the extracted PCF is saved as *My_Pipeline-extracted-pcf* in the same location along with the *My_Pipeline* file.
 - ★ IMPORTANT You must configure a **StartProgram** to store the extracted PCF. Otherwise, the software considers the final PCF as the extracted PCF.
- The **SmartPlant Foundation (*.xml)** target file type allows you to save drawing data and metadata to generate .xml files in either integrated and non-integrated environments that can be easily updated. For integration, you can use SmartPlant Interop Publisher to translate the generated .xml data file to a Smart Drawing available in SmartPlant Foundation. This option is available for **Isometric Drawings**, **Composed Drawings**, and **Volume Drawings** component types. For more information, see *Save as SmartPlant Foundation (*.xml)* (on page 76).
- 8. Click **OK** to save the files as specified.

■ NOTES

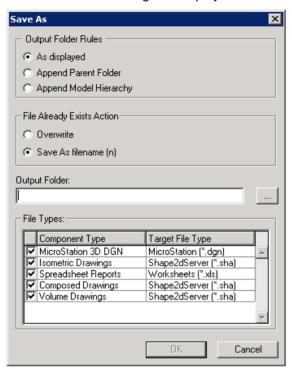
- The saved drawings retain the same names they had in this task.
- You can also extract all the sheets of a document to foreign formats such as MicroStation DGN, AutoCAD DWG and DXF files. A file is created for each sheet in the drawing with [drawing name]_[sheet name] as the filename. For example, if the drawing Volume_Drawing contains Sheet1 and Sheet2, two files are saved with the names Volume_Drawing_Sheet1 and Volume_Drawing_Sheet2.

Retrieve piping component file data (PCF)

When you create a Piping isometric document that uses the Iso_Stress style, the software does not create drawings. Instead it creates the data necessary for a Piping Component File (PCF). You can then output the PCF to the CAESAR II pipe stress analysis software.

To retrieve the PCF data, you use the Save As command.

1. Right-click the Piping Isometric document that uses the Iso_Stress style and select **Save As**. The **Save As** dialog box displays.



- 2. Specify the **Output Folder Rule** to be used. You can save the item as it displays in the **Console**, with its parent folder appended or with the entire model hierarchy appended.
- 3. Specify the **File Already Exists Action** to be used. This determines how you save the file if it has the same name of an existing file. Select **Overwrite** to replace the existing file, or select **Save As filename (n)** to save the file separately.
- 4. Specify the **Output Folder** location. Click **Browse** to display a dialog box to locate the appropriate folder location.
- 5. Check the **Isometric Drawings** component type. You can select multiple component types. For more information, see *Save As Dialog Box* (on page 78).

- 6. In the **Target File Type** list for the Isometric Drawing component type, select **PCF File** (.pcf).
- 7. Click **OK** to save the files as specified.

The PCF file is saved to the location you specified, ready for use in stress analysis.

■ NOTES

- The saved drawings retain the same names they had in this task.
- You can also view the PCF data with the View Extraction Data command. For more information, see View Isogen isometric extraction data (on page 114).

See Also

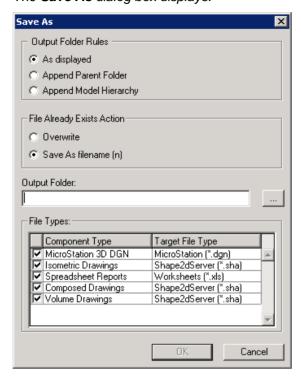
Save As Command (on page 72)
Isogen Isometric Drawings by Query (on page 99)
Isometric Drawing Styles (on page 125)

Save as SmartPlant Foundation (*.xml)

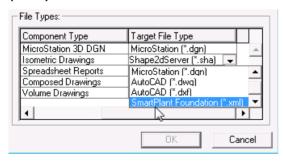
Use the **SmartPlant Foundation (*.xml)** target file type to save drawing data and metadata and generate .xml files in either integrated or non-integrated environments. This save option is available for **Isometric Drawings**, **Composed Drawings**, and **Volume Drawings** component types.

- Before you use the Save As command, you must define the Discipline property for your documents:
 - a. In the Console, right-click an item in the hierarchy, then select Properties on the shortcut menu.
 - b. Go to the WBS Tab.
 - c. Set the **Discipline** property. If your model has been registered using the SmartPlant Registration Wizard, this also adds the **Publish** command to the right-click menu for the selected document or documents. For a 3D Model Data document, the property is set to **SmartPlant Review Document**.

2. Right-click the Isometric, Volume, or Composed document, and select **Save As**. *The Save As dialog box displays.*



- 3. Specify the **Output Folder Rule** to use. You can save the item as it displays in the **Console**, with its parent folder appended or with the entire model hierarchy appended.
- 4. Specify the **File Already Exists Action** to be used. This determines how you save the file if it has the same name of an existing file. Select **Overwrite** to replace the existing file, or select **Save As filename (n)** to save the file separately.
- 5. Specify the **Output Folder** location. Click **Browse** to display a dialog box to locate the appropriate folder location.
- 6. Check the **Isometric Drawings, Composed Drawings, or Volume Drawings** component type. You can select multiple component types.
- 7. In the **Target File Type** menu list for the component type, select **SmartPlant Foundation** (*.xml).



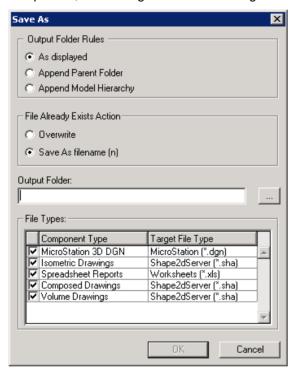
8. Click **OK** to save the files as specified.

The software generates an .sha file and two XML files in the specified location; an .xml data file and an .xml metadata file. The data .xml file is named ToolData_<Drawing Name>. The metadata file is named MetaData_<Drawing Name>.

You can now add the generated .sha in SmartPlant Interop Publisher to translate it to a Smart Drawing and publish the drawing to SmartPlant Foundation. SmartPlant Interop Publisher requires all generated files (.sha and .xml) to be located in the same folder location for translation and publishing. For more information about Smart Drawings and the translation and publish capabilities, refer to the SmartPlant Interop Publisher User's Guide.

Save As Dialog Box

Sets options for exporting drawings. You can open this dialog box by right-clicking a folder, component, or drawing and then selecting **Save As** on the right-click menu.



Output Folder Rules

Specifies how you want to save the hierarchy. The following options are provided:

- As displayed specifies the selection is saved as displayed.
- Append Parent Folder specifies the selected item is appended to the parent folder.
- Append Model Hierarchy specifies the selected item is appended to the hierarchy starting from the root and including the selected item.

File Already Exists Action

Specifies how you want to save the hierarchy if a file with the same name already exists.

- Overwrite overwrites the existing file.
- Save As filename (n) saves the file separately, and appends a number 'n' after the file name. For example, if the existing file name is FileName, the new file name is FileName (1).

Output Folder

Specifies the location to which to save the package.

Browse

Indicates a folder in which to save the drawings. You can select a local folder or a folder on another computer on the network.

File Type

Specifies the file formats to save for each drawing type. Check the box next to each drawing type you want to save. Use the **Target File Type** menu to specify the file type to which the drawing type is saved. The file types available for each drawing type are described in the following table.

Drawing Type	Target File Types
MicroStation	MicroStation J (V7 and V8) (*.dgn) 32MB limit
Isogen Isometric Drawings	Shape2DServer (*.sha)
■ NOTES	PCF file (*.pcf)
 For Isogen Isometric Drawings, the name of the drawing document becomes the prefix for all of the files. For example, if the drawing document name is My_Pipeline, the saved file names become: My_Pipeline.sha, My_Pipeline.pcf, and so on. For Isogen Isometric Drawings, a file is created for each sheet in the drawing with [drawing name]_[sheet name] 	Both (*.sha & *.pcf)
	All Files (*.*) - Includes all .sha and .pcf files, as well as all enabled supplementary files.
	MicroStation J (V7 and V8) (*.dgn)
as the filename. For example, if the drawing	AutoCAD (*.dxf)
My_Pipeline contains Sheet1 and Sheet2, two files will be saved with the names My_Pipeline_Sheet1 and	AutoCAD (*.dwg)
My_Pipeline_Sheet2.	SmartPlant Foundation (*.xml)
Spreadsheet Reports	Worksheets (*.xls)

Drawing Type	Target File Types
Composed Drawings, Volume Drawings, and Orthographic Drawings by Query	Shape2DServer (*.sha)
	MicroStation J (V7 and V8) (*.dgn)
	AutoCAD (*.dxf)
	AutoCAD (*.dwg)
Composed by Drawings and Volume Drawings	SmartPlant Foundation (*.xml)

See Also

Save As Command (on page 72)

Save Package Command

Saves the **Management Console** hierarchy from the selected component down. The package saves the setup information and any template definitions that exist on nodes within the selected hierarchy. You can access this command by right-clicking a folder or application component in the **Management Console**. You must have at least write permissions on the component to access the **Save Package** command.

★ IMPORTANT This command is not used in the normal workflow of creating drawings and reports. An administrator with write permissions to the SharedContent folders saves packages.

When a package is added back into the **Management Console**, it recreates the hierarchy that was saved with the package.

Packages are also used in the setup of a Drawings by Query Manager component for the creation of orthographic and Isogen isometric drawings.

Save Package Dialog Box (on page 81)

■ NOTES

- Output documents are not saved in a package.
- If the topmost component saved in the package is a folder, then the package can be placed under the model root or a folder. If the topmost component saved in the package is an application component, then the package can only be placed under a folder.
- When you place a package, the software adds all the components to the active permission group.
- To save drawings or reports externally, see Save As Command (on page 72).

Save a package

- Right-click a component in the Management Console hierarchy, and select Save Package
 The Save Package dialog box displays.
- 2. Specify a package name, package description, and tab name. For example, you can save a package named **Piping Isometric Drawing Package** to a new tab called **Iso**.
- Click **OK** to save the package.

The package is added to the **Add Component** dialog box on the specified tab. If a new tab name was specified, a new tab is added to the dialog box.

NOTE You can add the new package to the hierarchy by using the **New** command. Select the package on the **Add Component** dialog box. When you add a package, the software adds all the components to the active permission group.

See Also

Save Package Command (on page 81) Shortcut Menus (on page 34) Components Overview (on page 95) Isogen Isometric Drawings by Query Common Tasks (on page 101)

Save Package Dialog Box

Package Name

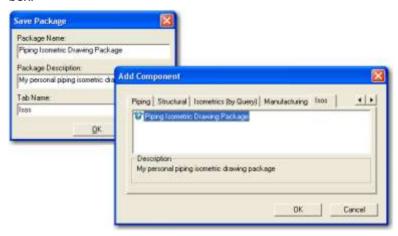
Specifies a name for the package.

Package Description

Describes the package.

Tab Name

Specifies the tab of the **Add Component** dialog box on which the package displays. You can pick an existing name or type a new tab name in this field. The next time you access the **Add Component** dialog box from an existing folder, the software adds the new tab and lists the new package on the tab. For more information, see *Add Component Dialog Box* (on page 40). For example, if you saved a package called Piping Isometric Drawing Package and added it to a new tab called Isos, an **Isos** tab is added to the **Add Component** dialog box:



See Also

Shortcut Menus (on page 34) Save a package (on page 81)

Updating Documents

Updating documents increases productivity because you can easily keep deliverables current. It is important to understand the different update capabilities.

■ NOTES

- You must install the SmartPlant Schema Component to update documents.
- If the drawing document you are looking at in the **Detail View** has a yellow icon (for example: ✓), the drawing document is a version 6.1 legacy Snapshot drawing. You should use the **Tools > Convert Legacy Snapshots** command to convert this document to a Composed Drawing for use in the current version of the software. If you do not convert the legacy snapshot drawing, you cannot update, revise, or publish the drawing.
- Any time you update a 3D model document, the software generates a single log file listing status information and any errors encountered during the process. The log file location is %TEMP%\EFUpdateCache\[3D Model ID]\[3D Model ID]\log.

Refreshing Document Contents

The **Refresh** command on the shortcut menu for a Console hierarchy item allows you to see which documents are out-of-date. For more information, see *Refresh* (*Shortcut Menu*) (on page 83).

Updating Documents Using Batch Processing

The **Batch** command queries the model to regenerate a single document or multiple documents. For volume and composed drawings, if you have edited the previous copy of the drawing (for example, by moving a label), the software remembers those changes and re-creates them. If batch processing is configured for the selected drawings, the update is performed on the Batch Server.

For more information, see Batch Processing - Intergraph Smart Batch Services (on page 283).

See Also

Refresh document status (on page 84) Print Command (on page 43)

Refresh (Shortcut Menu)

Compares the date of the last update of the document with the modification date in the model for any object that has a *positive* (can be seen) resymbolization in the drawing.

This command is not available until you generate documents. You can access this command by right-clicking an item in the **Management Console** hierarchy or in the **Detail View** and selecting **Refresh** on the shortcut menu. The software updates the status for all the expanded items within the parent node.

NOTE The **Refresh** command is not available for Spreadsheet Report documents. A Spreadsheet Report document is regenerated each time you update or print the report document. You can refresh the contents of a folder that contains reports.

For out-of-date documents \times , the command behavior implies the following:

- If the object is hidden entirely and is inside the drawing volume, but the style does not resymbolize the hidden lines, the object does not participate in the "out-of-date" definition.
- The modification date used for the object in the model can be for any property even if this property has no impact on the graphic. This means that a drawing could be considered out-of-date even though the graphic is up-to-date. For example, the approval status does not affect graphics. However, the view style you are using for your drawing can use a filter that sets approved objects to a specific color. A drawing document displays with an out-of-date definition because of a change to the approval status.
- Objects participating indirectly in the graphic as labels do not participate in the out-of-date definition. This means that, in rare cases, a label may be out-of-date on a drawing that is shown as up-to-date.

■ NOTES

- To refresh the Management Console to reflect changes made to loaded (expanded) tree view items, use the View > Refresh command.
- You can also use the Batch > Refresh command on the shortcut menu to perform your refreshes on the batch server. Batch > Refresh is not available for Spreadsheet Report documents.

Refresh document status

- 1. Right-click any item in the Management Console or in the Detail View.
- 2. On the shortcut menu, click **Refresh**. The software checks the model for any differences. The drawing icons change to reflect the status of the documents compared to the model.

NOTE After refreshing a folder or component, you can synchronize it with the model by right-clicking the item and then selecting **Update** (drawings by rule) or **Update Now** on the shortcut menu.

See Also

Refresh (Shortcut Menu) (on page 83) Icons for Components and Drawings (on page 23) Updating Documents (on page 82)

Update Now

Updates a single document or multiple drawing documents whether or not they are out-of-date. This command is available when you right-click on:

- A folder or component in the Management Console. All drawings in all components within the folder are updated.
- A component in the Management Console. All drawings in the component are updated.
- One or more documents in the **Detail View**. The selected drawings are updated.

This command works on your local computer regardless of the batch configuration.

For volume drawings, the **Update Now** command is not available until you place drawing volumes for a volume component in the Space Management task. For composed drawings, this command is not available until you create the drawings in a 3D task through **Tools** > **Drawing Console**. For reports, this command is not available until you create the report by choosing a report template.

■ NOTES

- If the software encounters a problem before or during the drawing update, it stops updating, displays either an error status or error message, and saves the errors to the log file. For more information, see *Conditional Drawing Update* (on page 91).
- The software preserves many of the modifications you make between regenerations of volume drawings. For example, if you annotate a volume drawing and then regenerate it, your annotations still display on the updated drawing.
- Documents created automatically in a Drawings by Rule component are automatically deleted by updating the component if the document no longer contains views.
- If the software cannot make a SmartPlant Foundation server connection when updating 3D Model Data documents, you are prompted to provide a valid login and password.
- If you place drawing property labels on a template, generate a drawing, move the labels on the drawing, and then update the drawing, the software remembers the new position of the labels on the drawing.

What do you want to do?

- Update all drawings in a folder or a component (on page 85)
- Update a report (on page 86)
- Update a single drawing (on page 87)
- Insert a note at a precise place on an isometric drawing (on page 87)

Update all drawings in a folder or a component

NOTE Before you update a folder or a component, you can refresh its documents to determine which documents are out-of-date. For more information, see *Refresh document status* (on page 84). You do not have to refresh before updating, but it can be helpful to determine which documents are out-of-date.

- 1. Right-click a folder or a component in the hierarchy. If you select a component, it must contain existing drawings or reports. If you select a folder, it must contain a component with existing drawings or reports.
- 2. On the shortcut menu, click Update Now.

The icons for the out-of-date documents change to show they are updated. If the Batch Server is configured, the command displays the **Drawing Batch** dialog box.

For more information, see *Batch Processing - Intergraph Smart Batch Services* (on page 283).

■ NOTE You can update an individual document by right-clicking the document and selecting **Update Now** on the shortcut menu.

Update a report

- Right-click a report document.
- On the shortcut menu, select Batch > Update to update the document on the batch server now or create a schedule to run the batch job. Alternatively, select Update Now to update locally.

The icon for the out-of-date document changes to show it is updated 🗸 .

3. If the batch server is configured, the $\textbf{Drawings Batch}\ \text{dialog box displays}.$

Batch Processing - Intergraph Smart Batch Services (on page 283)

NOTE When using the **Batch** > **Update** command and batch processing is configured for the selected document, the update is performed on the Batch Server. If batch processing is not configured, the command behaves the same as the **Update Now** command, performing a complete regeneration of the entire drawing on the local machine.

Update a single drawing

- 1. Right-click a document.
- On the shortcut menu, select Batch > Update to update the document on the batch server now or create a schedule to run the batch job. Alternatively, select Update Now to update locally.

The icon for the out-of-date document changes to show it is updated .

3. If the batch server is configured, the **Drawings Batch** dialog box displays.

**Batch Processing - Intergraph Smart Batch Services (on page 283)

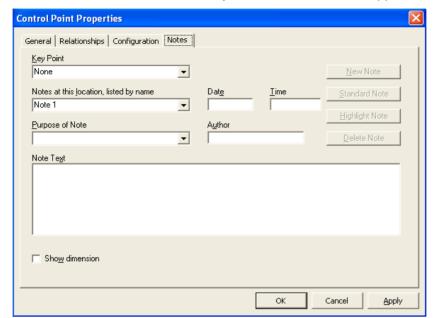
NOTE When using the **Batch** > **Update** command and batch processing is configured for the selected document, the update is performed on the Batch Server. If batch processing is not configured, the command behaves the same as the **Update Now** command, performing a complete regeneration of the entire drawing on the local machine.

Insert a note at a precise place on an isometric drawing

- 1. Select Insert > Control Point.
- 2. Select the pipe part in the model that needs a note.
 - TIP Use the guick pick tool to make the correct selection.



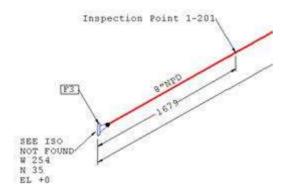
- 3. Position the control point on the centerline of the pipe part. For precision placement, use commands like **Measure**, **PinPoint**, or **Point Along**.
- 4. In the Type box, verify that Control Point is selected.
- 5. Confirm or change the option in the **Subtype** box on the ribbon.
- 6. In the Name box, define a name for the control point.
- 7. Click **Properties** 2.
- 8. Under the **Notes** tab, click **New Note**.
- 9. In the **Key point** box, select the control point to which to attach the note.
- 10. In the **Notes at this location, listed by name** box, type or select a name.
- 11. In the **Purpose of note** box, select **Fabrication** so that the note is picked up for inclusion in the drawing.
- 12. In the **Note text** box, type descriptive text for the note.



13. Check the **Show dimension** box if you want a dimension to appear on the drawing.

- 14. Click **OK**.
- 15. In the Drawings and Reports task, use the **Update Now** or **Batch** > **Update** commands to update the drawing that includes the pipe part with the associated control point.

The note appears where it was placed in the model.



Update

In marine mode or material handling mode, **Update** performs two functions:

- For existing drawings, this command updates a single document or multiple drawing documents.
- When drawings have not been created, this command generates and updates the drawings for the selected component. This option is only available for drawings-by-rule components.

NOTE Marine mode or material handling mode only: If you did not previously select the model contents for a drawings-by-rule component, the **Filter Properties for Asking Filter** dialog box displays, and you must select a filter. The dialog is the same as the **Filter Properties** dialog box, except that only the appropriate tabs for the asking filter are displayed. For more information, see Filter Properties Dialog Box.

For other plant mode drawings, this command updates a single document or multiple drawing documents.

Update is available when you right-click on:

- A folder or component in the Management Console. All drawings in all components within the folder are updated.
- A component in the Management Console. All drawings in the component are updated.
- One or more documents in the **Detail View**. The selected drawings are updated.

This command works on your local computer regardless of the batch configuration.

Update performs a smart update of a drawing. If a set of criteria is met, then an incremental update is performed only for the geometry of added, modified, and deleted objects. If the criteria are not met, then a full update of all geometry is performed.

All of the following criteria must be met for **Update** to trigger an incremental update:

- Each view in the drawing has had at least one full update previously performed.
- The only view properties that have changed are name and description. For more information, see View Tab (Drawing View Properties Dialog Box) (on page 349).
- The numbers of added, modified, or deleted objects in each view are less than a set percentage of the total number of objects.
- The software allows incremental updates for the view types in the drawing.

Any one of the following criteria cause **Update** to trigger a full update:

- A drawing view is new and has never had a full update.
- A drawing view is a section or detail view.
- A drawing view has been cropped.
- The view style, scale, orientation, or coordinate system of a drawing view have changed.
- A primary plate object in a view has been spit or unsplit.
- The numbers of added, modified, or deleted objects in each view are greater than a set percentage of the total number of objects.
- The software does not allow incremental updates for the view types in the drawing.

The **Update** command is not available until you create drawings in the **Management Console** using the **Create Drawing(s)** command. When a folder or component is selected, all views in the component(s) are updated, including unassigned views in the **Unassigned Folder** of the Drawing View Explorer.

■ NOTES

- If you have not previously selected a filter for the component, the **Select Filter** dialog box displays, and you must select a filter. For more information, see *Select Filter Dialog Box* (on page 108).
- An incremental update is usually faster than a full update, but still results in completely upto-date geometry for the view.
- Documents created automatically in a Drawings by Rule component are automatically deleted by updating the component if the document no longer contains views.
- The **Update** command is only available for drawings by rule. For other drawing types, the **Update Now** command is used. For more information, see *Update Now* (on page 85).

What do you want to do?

- Update all drawings in a folder or a component (on page 90)
- Update a single drawing (on page 91)
- Create automated major views for steel order scantling drawings
- Create an assembly drawing

Update all drawings in a folder or a component

NOTE Before you update a folder or a component, you can refresh its documents to determine which documents are out-of-date. For more information, see *Refresh document status* (on page 84). You do not have to refresh before updating, but it can be helpful to determine which documents are out-of-date.

- Right-click a drawings-by-rule folder or component in the hierarchy. The component must contain existing drawings or reports. The folder must contain a component with existing drawings or reports.
- 2. On the shortcut menu, click **Update**. The icons for the out-of-date documents change to show they are updated. If the Batch Server is configured, the command displays the **Schedule Wizard**.

Batch Schedule Wizard Common Tasks (on page 302)

■ NOTES

- Documents created automatically in a Drawings by Rule component are automatically deleted by updating the component if the document no longer contains views.
- You can update an individual document by right-clicking the document and selecting Update or Batch > Update on the shortcut menu.
- If you place drawing property labels on a template, generate a drawing, move the labels on the drawing, and then update the drawing, the software remembers the new position of the labels on the drawing.

Update a single drawing

- 1. Right-click a drawings-by-rule document.
- 2. On the shortcut menu, select **Batch** > **Update** to update the document on the batch server now or create a schedule to run the batch job. Select **Update** to update locally. The icon for the out-of-date document changes to show it is updated .
- 3. If the batch server is configured, the **Schedule Wizard** displays.

Batch Schedule Wizard Common Tasks (on page 302)

■ NOTES

- When using the Batch > Update command and batch processing is configured for the selected document, the update is performed on the Batch Server. If batch processing is not configured, the command behaves the same as the Update command, performing a complete regeneration of the entire drawing on the local machine.
- If you place drawing property labels on a template, generate a drawing, move the labels on the drawing, and then update the drawing, the software remembers the new position of the labels on the drawing.

Conditional Drawing Update

Before updating a drawing, Smart 3D ensures necessary information is present before changing the existing drawing document. If any of the following conditions are true before the update process begins, the software displays an error message that lists the missing items, and makes no changes to the drawing. If any of the following conditions are true after the update process

begins, the drawing update process stops, changes to an error status (.), and Smart 3D preserves the previous state of the drawing document to avoid any data loss.

Reference Data Conditions

- View style cannot be found.
 - Graphic preparation rule cannot be found.
 - Graphic rule cannot be found.
 - Label rule cannot be found.
 - Dimension rule cannot be found.
 - North arrow rule cannot be found.
 - Matchline rule cannot be found.
 - View rule cannot be found.
- View style filter cannot be found.
- Emptyvw.sha file cannot be found.
- Styles.sha file cannot be found.

Model Data Conditions

- Associated volume cannot be found.
- Associated coordinate system cannot be found.

Memory Conditions

- Not enough available memory to begin the update process.
- Not enough available memory to complete the update process after it has started.

■ NOTES

- Failure conditions found before the update process can be viewed in the **Drawings.log**.
- Failure conditions found during the update process can be viewed when you right-click a drawing and select View Log.
- Failure conditions are not logged when the **Update** command is used on a drawing view.
 You must use the **Update Now** command in either the **Drawing Console** or Drawings and Reports task to save the error(s) to the log.
- If an orphan view (a view that exists in the model database without a corresponding view) exists, it is removed from the database and a description of the error is logged in the Drawings log file in <Temp Folder>\Logs.

Restore

Restores a drawing document from a model restored from a backup. This command is available when you right-click on a document in the **Drawing Console** or the **Drawings and Reports** task.

■ NOTES

- When a document is restored, all document properties are overwritten.
- You cannot restore a document if the document is created in your current model after the backup is created. If you select a group of documents to restore, new documents in the group are not restored. All other documents are restored from the backup model.
- Views that are created in your current drawing after the backup is created are lost when you select **Restore**.

What do you want to do?

- Create a backup to use for restoration (on page 93)
- Restore a document from a backup model (on page 93)

Create a backup to use for restoration

- In the Project Management environment, create a backup file of your current model using Tools > Backup. For more information, see Backup in the Project Management User's Guide.
- 2. Create a new model in the site by restoring the backup file using **Tools** > **Restore**. For more information, see *Restore* and *Restore Wizard* in the *Project Management User's Guide*.
- TIP Give the new model a name that makes it easy to identify as the backup of your current model.

Restore a document from a backup model

- In the **Drawing Console** or the **Drawings and Reports** task, right-click on a drawing document in your current model, and select **Restore**.
 - The Restore Document dialog box displays.
- 2. Select the backup model from the **Model** box, and then click **OK**.
 - The document in your current model is replaced with the document from the backup model.

View Log Command

Displays the log information for the selected drawing. To access this command:

Right-click a drawing document at any time, and select View Log on the shortcut menu.

For drawings by rule, right-click a component after using Create Drawing(s) or Update at the component level, and select View Log on the shortcut menu. View Log is only available if there was an error during the create or update operations.

SECTION 6

Components Overview

Several specialized components allow you to access commands for configuring templates and generating drawings and reports. The various types of components can be divided into two groups: application components and folder components. Many of the application components correspond to specific types of drawings, such as volume drawings and composed drawings. The Spreadsheet Reports component provides access to report-related commands. You organize drawings and reports in folders. You can add folders to the root and to other folders. Each component has a different icon and right-click menu.

You can copy and paste components with some restrictions. For example, application components cannot contain folders or other application components. If you copy a folder, you can paste it under a folder but not under another component type. If you copy a Volume Drawing component, you can paste it under a folder. If you copy other types of components, you can paste them under folders but not under other types of components.

Your administrator can assign permissions to the different components using commands in the Project Management task. For example, the administrator can set permissions so that only the piping designers have write privileges on Isogen Isometric Drawings. For more information, see the *Project Management User's Guide*.

There are several types of delivered components. Their names reflect the type of drawing or report they create. When you right-click the root or a folder and select **New**, the **Add Component** dialog box displays. The dialog box includes a **General Tab** for general types of drawings or reports and additional task-specific tabs with delivered folders and packages. For more information, see *Add Component Dialog Box* (on page 40).

NOTE You can also create and manage components from the 3D modeling tasks by using the **Tools > Drawing Console** command. For more information, see the *Common User's Guide*.

The following components are shown on the **General** tab of the **Add Component** dialog box. Other components are shown on other tabs.

Delivered Component

3D Model By Query

Description

Creates a 3D Model By Query component in the Console. You can use the 3D Model By Query component to export 3D model data in bulk as CAD (SAT) files. The software uses a filter-based query to collect the objects and document them automatically. More information is available in the Orthographic Drawings User's Guide.

Delivered Component

Description

3D Model Data

Creates a 3D Model Data component in the Console. Right-click the component and select **Setup** on the shortcut menu to specify a filter that identifies the objects you want collected by the 3D Model Data component. You can use the 3D Model Data component to output SmartPlant Review files or CAD (SAT) files. More information is available in the Orthographic Drawings User's Guide.

MicroStation 3D DGN

Creates a MicroStation 3D DGN component in the Console. Right-click the component and select Setup from the shortcut menu to define the MicroStation seed file and style to use in generating the component documents. More information is available in the Orthographic Drawings User's Guide.

Folder

Creates an empty folder in the Console.

Drawings By Query Manager

Creates a Drawings by Query Manager in the Console. The Drawings by Query Manager is used in conjunction with other components, such as the Orthographic Drawing by Query and Isogen Isometric Drawing by Query components, to complete the query for objects in the model. The Drawings by Query Manager provides the filter that specifies the "where" side of the query. It tells the query "where" to look for the objects specified by the component "what" filter. More information is available in the Orthographic Drawings Users Guide and the Isogen Isometric Drawings User's Guide.

Generic Module Folder

Creates a Generic Module Folder in the Console. The Generic Module Folder component provides a way for you to run your custom VB modules to create custom drawings. You set up the Generic Module Folder component to use your custom VB module. More information is available in the Orthographic Drawings User's Guide.

🔃 Imported Folder

Imports an external Windows folder from a shared network folder into the Management or Drawing console. The imported folder can contain any type of file available in Windows. You can manage the imported documents, and publish them using Imported Folder menu options. For more information, refer to Drawings Folders in the Orthographic Drawings User's Guide.

Delivered Component

on voice compensi.

Description

Composed Drawings

Creates a Composed Drawing component in the Console. Composed drawings are orthographic drawings created in a 3D task such as Common. The composed drawing component manages the composed drawings you create. More information is available in the *Orthographic Drawings User's Guide*.

Orthographic Drawings by Query

Creates an Orthographic Drawings by Query component in the Console. The Orthographic Drawing by Query component allows you to create drawings for many objects in the model all in the same manner. This component does not require physical volumes in the model. The software uses a filter-based query to collect the objects and document them automatically. More information is available in the *Orthographic Drawings User's Guide*.

Isogen Isometric Drawings by Query

Creates an Isogen Isometric Drawings by Query component in the Console. You create an isometric drawing by associating an Isogen Isometric Drawing by Query component to a Query Manager. The Isogen Isometric Drawing by Query component specifies the "what" portion of the query, while the Query Manager specifies the "where." More information is available in the *Isogen Isometric Drawings User's Guide*.

Spreadsheet Reports

Creates a Spreadsheet Report component in the Console. More information is available in the *Reports User's Guide*.

Search Folder

Creates a Search Folder in the Console. Search Folders allow you to search for documents based on common properties such as out-of-date status, approval, or documents that have been published to a certain contract in integrated environment. More information is available in the *Orthographic Drawings User's Guide*.

5 Volume Drawings

Creates a Volume Drawings component in the Console. The Volume Drawing component uses a template to create drawings. You can place a view on the template and associate the view with a view style to control the output. To define the contents of the view, you create a drawing volume in the model. More information is available in the *Orthographic Drawings User's Guide*.

For drawing components used in marine mode, see *Drawings by Rule Components* in the *Orthographic Drawings User's Guide*. You can access this document using the **Help > Printable Guides** command in the software.

See Also

Shortcut Menus (on page 34)
Icons for Components and Drawings (on page 23)
New Command (on page 38)
Delivered Drawing Types (on page 20)

SECTION 7

Isogen Isometric Drawings by Query

Isometric drawings communicate several important types of information to a fabrication workshop. In the case of a piping isometric drawing, this information includes pipe cut lengths, bend angles, and welds. You create an isometric drawing by associating an Isogen Isometric Drawing by Query component to a Query Manager. The Isogen Isometric Drawing by Query component specifies the *what* portion of the query, while the Query Manager specifies the *where* portion.

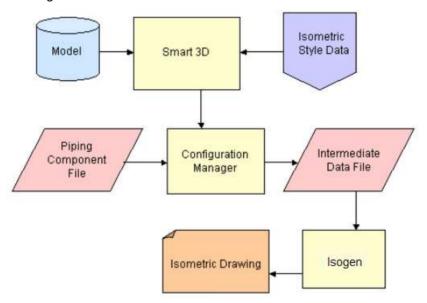
The basic workflow for creating Isogen Isometric Drawings by Query is as follows:

- Create an Isogen Isometric Drawing by Query component that specifies what you want to document.
- Create a Drawings by Query Manager that specifies where to look for the data.
- Run the filter-based query.
- Create the drawings.
- Update the drawings, if needed.
- Publish the drawings to a viewable graphic file; no physical data is published.

Each Isogen Isometric Drawing by Query component has an associated isometric style. The delivered styles are Iso_Pipeline, Iso_Piperun, Iso_PenSpool, Iso_Spool, Iso_WBS, Iso_Stress, Iso_Assembly, Iso_System, Iso_HVAC, and Iso_CableTray. You use **Isogen Configuration** to set options for isometric drawing format and content. You can open **Isogen Configuration** from any of the isometric drawing styles by right-clicking the drawing style in the **Management Console** or **Drawing Console**, and selecting **Edit Options** on the shortcut menu. For more information, see *Isogen Configuration* (on page 136).

After creating an Isogen isometric drawing, you can open it for viewing, editing, printing, and publishing (if your model has been registered using the **SmartPlant Registration Wizard**). To troubleshoot a drawing, you can use the **View Data** command to access part and reports information and log files.

The software uses Isogen, a third-party engine made by Alias, to generate isometric drawings from the three-dimensional model. The following illustration shows the process of creating the drawings.



The application creates a piping component file (PCF), which is used along with the isometric style information to create an intermediate data file (IDF). The Isogen engine then reads the IDF and creates the isometric drawings.

Administrator Setup

Your administrator should set up appropriate isometric styles to use with the Isogen Isometric Drawings by Query component. The administrator is also responsible for creating filters that define what to look for and filters that specify where in the model to look for the objects. The template and the *what* filter information within the component are saved as a package.

Drawings by Query Manager

The Drawings by Query Manager uses the filter to specify where to look for the objects included in the drawing.

For more information on the filters necessary for setting up a Drawings by Query component, see *Drawings by Query Filters* (on page 102).

Isogen Isometric Drawings by Query Common Tasks

You use the following tasks when you create Isogen Isometric Drawings by Query.

For information on the filters required for creating Isogen Isometric Drawings by Query, see *Drawings by Query Filters* (on page 102).

Setup an Isogen Isometric Drawing by Query Component

Create and perform set up for an Isogen Drawing by Query component. For more information, see *Set up an Isogen Isometric Drawing by Query component* (on page 105). When you select a filter in **Setup**, you are specifying the "what" portion of the query. In other words, you are specifying the objects to be included in the drawing.

Save the Isogen Isometric Drawings by Query Component as a Package

Save the Isogen Isometric Drawing by Query component as a package, so you can join it with a Query Manager to generate the query drawings. For more information, see *Save a package* (on page 81).

Add a Drawings by Query Manager

Create a Drawings by Query Manager, then set it up to specify the "where" portion of the query for the Isogen Isometric Drawing by Query component. For more information, see *Set up a Drawings by Query Manager component* (on page 110).

Run the Query

Use the **Run Query** command to execute the query specified by the Isogen Isometric Drawing by Query component and the Drawings by Query Manager. For more information, see *Run Query (Shortcut Menu)* (on page 111).

Create or Update the Drawings

To create or update the drawings, right-click the component, and then select the appropriate command. For more information, see *Updating Documents* (on page 82).

View the Drawing Log

You can view the drawing log to see any messages associated with the drawing. For more information, see *View Log Command* (on page 93).

Set Drawing Properties

Specify the properties for the drawing component or drawing documents by right-clicking and selecting **Properties**. For more information, see *Edit document properties* (on page 47).

View Extraction Data

View the extraction data by right-clicking an isometric drawing and selecting **View Extraction Data** on the shortcut menu. For more information, see *View Isogen isometric extraction data* (on page 114).

Compare 2D Drawing Objects to 3D Model Objects

You can open an Isogen Isometric Drawing in a 3D task, and then compare the 2D drawing

objects in the drawing document to 3D model objects. For more information, see *Compare 2D Drawing Object to 3D Model Object* (on page 444).

Publish the Isogen Isometric Documents

Publish the Isogen Isometric Drawing by Query documents. You can publish only if your model has been registered using the **SmartPlant Registration Wizard.** For more information, see *Publishing Documents* (on page 517).

NOTE The viewable files created when you publish drawings and reports provide relationship links to the 3D Model Data. You must also publish the 3D model data to provide the navigation between the viewable files and the 3D model data. For more information, see the *Orthographic Drawings User's Guide* available from **Help > Printable Guides**.

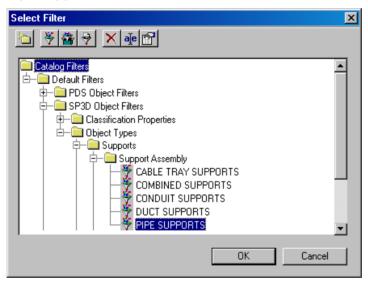
See Also

Isometric Drawing Styles (on page 125)

Drawings by Query Filters

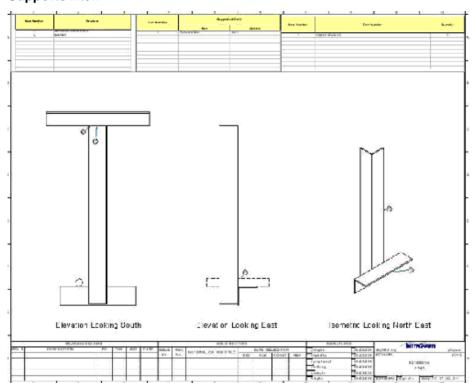
Filters determine the contents of the documents produced by Drawings by Query components (3D Model By Query, Orthographic, Isogen Isometric, and Drawings by Query Manager). For Orthographic and Isogen Isometric Drawings by Query components, the **Setup** command specifies a *what* filter that access the model database and determines which objects are included in the drawings, as well as which hierarchy is traversed to create a tree in the **Management Console**. **Setup** then defines a Drawings by Query Manager component to specify a *where* filter that determines the position in the hierarchy and, therefore, the location of the objects included in the drawing.

For example, if you are creating Orthographic drawings that include pipe supports, you create a new Orthographic Drawings by Query component, and then run **Setup** on the component to select a normal filter that returns only pipe support objects. For example, you might choose the **Pipe Supports** filter as shown below:



After saving your Orthographic Drawings by Query component setup as a package, you then define a Drawings by Query Manager component to determine *where* in the model to collect the objects you specified in the *what* filter. When you run **Setup** on the Drawings by Query Manager component, you select a normal filter.

When you run the query, the software compounds the *where* filter with the *what* filter to return the needed objects. Each object is documented in a drawing using the template and rules that



you setup for the package. The following graphic shows an example drawing using the **Pipe Supports** filter:

For more information on defining filters, see the *Common User's Guide* available from **Help > Printable Guides**.

- **NOTE** A *what* filter can specify non-graphical objects, such as various system nodes in the **System** tab, folders in the **Space** tab, WBS projects and items in the **WBS** tab, and Reference 3D Model nodes in the **Reference 3D** tab of the **Workspace Explorer**. When you specify non-graphical objects in the *what* filter, you must also specify a Navigator Rule in the **Drawing View Properties** dialog box.
- **TIP** You can use the **SystemRangeNavigator.dll** Navigator Rule to include all graphical objects under a non-graphical object. For more information on navigation rules, see the *Navigator Rules* section in Orthographic Drawings by Query.

See Also

Isogen Isometric Drawings by Query Common Tasks (on page 101)

Setup (Isogen Isometric Drawing by Query Component Shortcut Menu)

Sets component options for creating Isogen Isometric Drawings by Query. This command is available on the shortcut menu for the Isogen Isometric Drawing by Query component.

Setup Dialog Box (Isogen Isometric Drawing by Query Component) (on page 106)

What do you want to do?

- Set up an Isogen Isometric Drawing by Query component (on page 105)
- Create an isometric drawing (on page 112)

Set up an Isogen Isometric Drawing by Query component

1. Right-click the folder where the new Isogen Isometric Drawing by Query component are to be stored, then select **New**.

The Add Component dialog box displays.

- 2. Select the Isogen Isometric Drawings by Query component, and then click ${\bf OK}.$
 - The software creates the Isogen Isometric Drawing by Query component in the folder.
- 3. Right-click the Isogen Isometric Drawing by Query component, and then select **Setup** on the shortcut menu.
 - The **Setup** dialog box displays.
- 4. Specify a filter in the Filter field. The list shows the most recently selected filters. Click More to display the Select Filter dialog box and specify a filter. The filter you select is the what portion of the query, such as Pipe Runs. Click Properties of to display the current filter properties. For more information on filters for Drawings by Query, see Drawings by Query Filters (on page 102).
 - **NOTE** Select a filter that is appropriate for the isometric drawing styles to be used for this Isogen Isometric Drawing by Query component. For example, for the Iso_Pipeline style, the filter must contain piping and pipeline systems. For the Iso_Spool and Iso_PenSpool styles, the filter must contain spool assemblies. For the Iso_CableTray and Iso_HVAC styles, the filter must contain cableways and HVAC runs.
- 5. In the first Styles list, select the type of isometric drawing to produce. You can select Piping, HVAC, or Cabletray. Selecting the type of isometric drawing causes the software to filter the styles in the list below. Next, specify the style to use. Click More in the list to display a list of all available styles. For more information, see Select Drawing Style Dialog Box (on page 106).

- 6. Click **Properties** to display **Isogen Configuration**. Modify options for the isometric styles you added as needed. For more information, see *Customize isometric style options* (on page 171).
- 7. Click **OK** to create the Isogen Isometric Drawing component as specified.

To use the component to generate isometric drawings, you must save it as a package, and then associate the package to a Drawings by Query Manager component. For more information, see *Set up a Drawings by Query Manager component* (on page 110).

■ NOTES

- To delete a component, right-click the component, and then select **Delete**.
- To rename a component, right-click the component, and then select Rename.

Setup Dialog Box (Isogen Isometric Drawing by Query Component)

Sets options on Isogen Isometric Drawing components.

Filter

Identifies the filter to use to define the *what* portion of the query. The software uses the filter to determine the objects included in the drawings when they are generated. Select **More** in the **Filter** list to display the **Select Filter** dialog box. Click **Properties** of to display the **Filter Properties** dialog box to edit the filter as needed. For more information on filters for Isogen Isometric Drawings by Query, see *Drawings by Query Filters* (on page 102).

Style

Specifies the type of isometric drawing to create and the isometric drawing style to use when generating the output for the isometric drawing. The software enables you to create the following types of isometric drawings: **Piping**, **HVAC**, and **Cabletray**. The delivered styles include Iso_Pipeline, Iso_Piperun, Iso_Spool, Iso_PenSpool, Iso_WBS, Iso_Stress, Iso-Assembly, Iso_System, Iso_HVAC, and Iso_CableTray. Your administrator may provide more isometric drawing styles. Click **More** in the list to display the **Select Drawing Style** dialog box. For more information, see *Select Drawing Style Dialog Box* (on page 106).

▶ NOTE The isometric drawing styles that the software displays depend on the isometric drawing type that you have selected. For example, if you select HVAC, only the HVAC-specific isometric drawing styles (Iso HVAC) display in the list.

You can click **Properties** open **Isogen Configuration**, and edit the isometric style as needed. For more information, see Isogen Configuration.

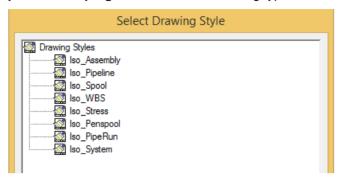
See Also

Setup (Isogen Isometric Drawing by Query Component Shortcut Menu) (on page 104)

Select Drawing Style Dialog Box

Displays a list of all available isometric styles for the selected drawing type This dialog box displays when you select **More** in the **Style** list on the **Setup** dialog box for an Isogen Isometric

Drawing component or package. The example below shows the styles that are available when you select **Piping** as the isometric drawing type.



See Also

Setup (Isogen Isometric Drawing by Query Component Shortcut Menu) (on page 104) Setup Dialog Box (Isogen Isometric Drawing by Query Component) (on page 106)

Select Filter

Specifies a filter for orthographic or Isogen isometric drawings created by Drawings by Query components. The filter narrows the objects returned for the drawings.

For more information on setting up filters for Drawings by Query components, see *Drawings by* Query Filters (on page 102).

Select Filter Dialog Box (on page 108)

Select Filter Dialog Box

Creates, edits, deletes, and selects filters for use with the **Define Workspace**, **Surface Style** Rules, and other Select by Filter commands, including Project Management's Model Data Reuse (MDR), Drawings View Styles, and Reports commands that require runtime filter selection. You can access this dialog box in several ways.

- Select File > Define Workspace, and select the More option in the Filter box.
- Select Format > Surface Style Rules, click New or Modify, and then select the More option in the Filter box.
- Select Tools > Select by Filter.

The tree view displays the following types of filters:

- Catalog Filters These filters are used to reference data in the Catalog. For example, a catalog filter could apply to company-wide operations. Your administrator can define Company_Filter_1, Company_Filter_2, and so forth.
- Model Filters These filters are available to everyone assigned to a specific model database. There are delivered catalog filters to query on the different types of model objects. You must have the appropriate privileges to create, edit, or delete these filters.
- My Filters These are personal filters that you create and place in the My Filters folder. They are visible only to you, the owner. You cannot see the personal filters of others, and they cannot see your personal filters. Select a filter from one of the listed filters, or create a new filter to meet your specific requirements.



New Folder

Creates a new folder.

New Filter (Simple or Asking)

Displays the New Filter Properties dialog box so that you can create a new filter. Asking filters allow you to specify the parameters of the search. An asking filter has built-in functionality to ask for values (with boxes that you are required to supply). The values apply to properties that you have already designated you will supply when the filter runs. Asking filters are portable between models.

NOTE Model Data Reuse (MDR) does not support asking filters. The only valid filter types for an MDR transaction are System, Permission Group, Object Type, Volume and Properties. You can define the filter on any one of these tabs or in a combination using multiple tabs.



New Compound Filter

Displays the New Compound Filter Properties dialog box, which you use to create a new compound filter containing the Or, And, or Not operators. Compound filters are not supported for MDR.



New SQL Filter

Displays the New SQL Filter Properties dialog box, in which you can type the text of an SQL query. SQL filters are not supported for MDR

× Delete

Removes a filter or folder from the Select Filter list. If you delete a folder, the software also deletes its contents.

Rename

Changes the name of an existing filter or folder from the **Select Filter** list.



Properties

Displays the Filter Properties dialog box so that you can select the properties that determine your filter search criteria.

■ NOTES

- If this dialog box is activated from the **Select by Filter** command, you can select multiple filters on this dialog box. Hold CTRL or SHIFT, and click each filter. When you click OK, all objects that fit the selected filters are selected.
- If this dialog box is activated from the Select by Filter command, it clears the select set before adding objects to the select set.

Setup (Drawings by Query Manager Component Shortcut Menu)

Sets options for creating a Drawings by Query Manager component. This command is available on the shortcut menu when you right-click a Drawings by Query Manager component.

The Drawings by Query Manager component is used in conjunction with other components, such as the Orthographic and Isogen Isometric Drawing by Query components, to complete the query for objects in the model. The Drawings by Query Manager provides the filter that specifies the *where* side of the query. It tells the query *where* to look for the objects specified by the component *what* filter.

Setup Dialog Box (Drawings by Query Manager Component) (on page 111)

Set up a Drawings by Query Manager component

The Drawings by Query Manager component works in conjunction with the Orthographic Drawing by Query and Isogen Isometric Drawing by Query components. Before using this command, you must create packages for your Orthographic Drawing by Query and Isogen Isometric Drawing by Query components. For more information, see *Save Package Command* (on page 81).

1. Right-click the folder in which to create your Drawings by Query Manager.

The software displays the Add Component dialog box.

- TIP You can store the Drawings by Query Manager anywhere in the **Console**, but it is best to store it in the same location as the components with which it works.
- 2. Select the Drawings by Query Manager component, and then click **OK**.
 - The software creates the Drawings by Query Manager component in the folder.
- 3. Right-click the Drawings by Query Manager component, and then select **Setup** to specify the properties for the component.
 - The software displays **Setup** dialog box.
- 4. Specify a filter in the Filter field. The list shows the most recently selected filters. Select More in the list to display the Select Filter dialog box, and then specify a filter. Click Properties to display the current filter properties.
 - **NOTE** The filter that you select is the *where* portion of the query, as opposed to the *what* portion specified when you set up the Orthographic Drawing by Query or Isogen Isometric Drawing by Query component. The filter that you specify here tells where in the model you want to look for the objects. For more information on filters for Drawings by Query, see *Drawings by Query Filters* (on page 102).

- 5. In the **Package** field, specify the Orthographic Drawing by Query or Isogen Isometric Drawing by Query package that you created. The list contains the most recently selected packages. Select **More** to display the **Select Package** dialog box. For example, if you are defining a Drawings by Query Manager for an Orthographic Drawing by Query, select an Orthographic Drawing by Query package.
- 6. Click **OK** to save the settings.

To create the drawings, you need to run the query. For more information, see *Run Query (Shortcut Menu)* (on page 111).

■ NOTES

- To delete a component, right-click the component, and then select **Delete**.
- To rename a component, right-click the component, and then select Rename.

Setup Dialog Box (Drawings by Query Manager Component)

Sets options for creating Drawings by Query Manager components.

NOTE You must create an Orthographic Drawing by Query, Isogen Isometric Drawing by Query, or 3D Model by Query package before setting up the Drawings by Query Manager component. For more information, see *Save Package Command* (on page 81).

Filter

Identifies the filter to use to define the *where* portion of the query. The software uses the filter to determine where to look for the objects requested in Orthographic Drawing by Query and Isogen Isometric Drawing by Query components when they are generated. Select **More** in the list to display the **Select Filter** dialog box. Click **Properties** of to display the current filter properties. For more information on filters for the Drawings by Query Manager, see *Drawings by Query Filters* (on page 102).

Package

Specifies the package to use in completing the query.

See Also

Setup (Drawings by Query Manager Component Shortcut Menu) (on page 109)

Run Query (Shortcut Menu)

Runs the query associated with the selected Drawings by Query Manager component. For example, if you execute **Run Query** on an Isogen Isometric Drawing by Query package associated with a Drawings by Query Manager, the software looks for piping in the model. The query results display beneath the style in the **Management Console**. You can create isometric drawings from the query results by right-clicking on the component and selecting **Create Drawing(s)**.

If you have an Orthographic Drawing by Query package associated to a Drawings by Query Manager, the **Run Query** command runs the query associated with the components, collects the objects from the database, and builds the information that will be included in the drawing. You can then create orthographic drawings from the query results by running the **Create Drawing(s)** command on the Orthographic Drawing component.

The Run Query command uses the filters specified when you performed Setup components.

■ NOTES

- When working in a Global Workshare Configuration with users logging into both Host and Satellite systems for more than one site, you can encounter problems with the filters defined for a Drawing by Query package. For example, if you create a Filter Root Folder for a particular site, and then you try to run queries for drawings associated to this filter from another site, the Run Query command is unable to update the drawings due to a lack of permission against the Filter Root Folder. You have to move or transfer the Filter Root Folder to the appropriate site where the Run Query command is executed.
- If the selected package was created and added manually, the folder name for the package must match the value for **pkgid** in the XML file. Edit the name of the folder to match the **pkgid** value.
- ★ IMPORTANT In marine mode, this command displays in the Ship Root and Folder shortcut menus for drawings by rule, but is not used with drawing by rule components.

See Also

Create Drawing(s) Command (on page 36)

Create an isometric drawing

- 1. Verify that at least one isometric piping drawing exists in the **Console** hierarchy. If none exists, add a Drawings by Query Manager component for isometric drawings.
- 2. Right-click the Drawings by Query Manager component, and then select **Setup** on the shortcut menu.
- 3. On the **Setup** dialog box, specify a filter in the **Filter** field. The list displays the most recently selected filters. Click **More** to display the **Select Filter** dialog box, and then specify a filter.
- 4. Specify a **Package** to use for the isometric drawings, and then click **OK**.
- 5. Right-click the isometric drawing style, and then select **Run Query**.

The hierarchy updates with the available items for that style.

- TIP To display available items for all isometric styles in a component, right-click the component in the **Console**, and then select **Run Query**.
- 6. Right-click a folder, component, pipeline, or spool, and then select **Update Document(s)** to create isometric drawings.

TIP You can extract multiple objects (for example, pipelines) if you hold **Ctrl** or **Shift** while you select the objects in the **Console** or **Detail View**.

NOTE You can remove items from an isometric piping drawing component by modifying the filter and running the query again.

See Also

Isogen Isometric Drawings by Query (on page 99)

View Extraction Data

Helps you troubleshoot extraction errors by displaying part and report information for a line that has been processed. You can view the log file and Piping Component File (PCF) data for the extraction. If you have customized the style to produce report files such as the neutral file and cut pipe list, you can view those files directly in the **Extraction Data** dialog box. If you have overridden any style settings for an Isogen isometric drawing, you can view the style fragment data in the **Extraction Data** dialog box as well. For more information about overriding style options, see *Option Override (Isogen Isometric Drawing Shortcut Menu)* (on page 196).

This command is available on the shortcut menu by right-clicking a line or isometric drawing in the tree view or list view.

Saving Extraction Data to File

If you need to review the data in more detail or the extraction data files are too large to view in the **Extraction Data** dialog box, you can right-click the isometric drawing document, and then select **Save As** to save the extraction information to a file. On the **Save As** dialog box, select the **Isometric Drawings** component type, then specify **All Files** in the **Target File Type** list to save all of the extraction data files. You can also specify individual files. For example, specify the **Target File Type** as **PCF File** to only save the PCF file information. For more information, see *Save As Command* (on page 72).

You can also save the original extracted PCF that is generated from Smart 3D in addition to the final PCF. Use the **Drawing Manager > Pre and Post Processors > Settings > Pre-processor** option to configure an Isogen preprocessor. After updating the drawing, when you use **Save As** with the **All Files** option selected, the original extracted PCF file from Smart 3D is saved as *<filename>-extracted-pcf* in addition to the final PCF supplied to Isogen. For example, if the final PCF file name is *My_Pipeline*, then the extracted PCF is saved as *My_Pipeline-extracted-pcf*.

★ IMPORTANT You must configure an Isogen preprocessor to store the extracted PCF. Otherwise, the software considers the final PCF as the extracted PCF.

Extraction Data Dialog Box (on page 116)

What do you want to do?

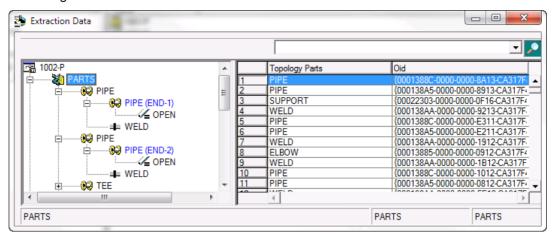
- View Isogen isometric extraction data (on page 114)
- View style fragment data (on page 115)

View Isogen isometric extraction data

- 1. In the **Console** hierarchy, right-click an Isogen isometric drawing.
- 2. Select View Extraction Data on the shortcut menu.

The dialog box displays the part information and report information for the line, including the log file and Piping Component File (PCF) data.

3. To view the extracted parts information, select and expand the **Parts** node on the left side of the dialog box.



4. To view the report information, select and expand the **Reports** node on the left side of the dialog box.



5. Type a query into the **Search** field at the top of the dialog box, and press ENTER or click to locate the parts in the extraction data. For more information, see *Extraction Data Dialog Box* (on page 116).

■ NOTES

- If you have customized the style to produce report files, such as the neutral file and cut pipe list, you can also view those files directly in the dialog box.
- If you need to review the data in more detail or the extraction data files are too large to view in the Extraction Data dialog box, you can right-click the isometric drawing document and select Save As to save the extraction information to a file.

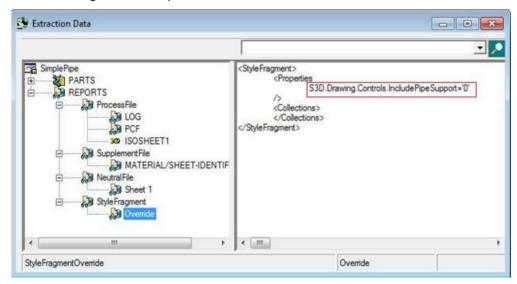
View style fragment data

1. Right-click an Isogen isometric drawing, and then select View Extraction Data.

The software displays the Extraction Data dialog box.

2. Expand the Reports and StyleFragment nodes, and then select Override.

The contents of the style fragment, which includes the override setting, display in the view area on the right. An example is shown below.



NOTE For more information about overriding style options, see *Option Override (Isogen Isometric Drawing Shortcut Menu)* (on page 196).

Extraction Data Dialog Box

Provides troubleshooting information for the isometric drawing extraction process by displaying parts and reports. You can access this dialog box by right-clicking an Isogen isometric drawing and then selecting **View Extraction Data** on the shortcut menu.

NOTE If you need to review the data in more detail or the extraction data files are too large to view in the **Extraction Data** dialog box, you can right-click the isometric drawing document and select **Save As** to save the extraction information to a file. For more information, see *Save to a file* (on page 73).

Search Box

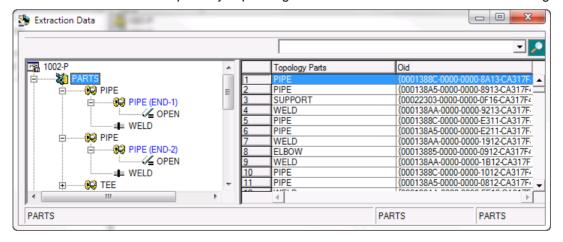
The **Search** box at the top of this dialog box allows you to find parts that meet the search criteria. You must type the search text as *?<command>:<value>*. The table below lists the allowable commands.

Command	Value	Action
COORD	Type a space separated coordinate value	 Displays a list of parts at the specified coordinate. Clicking the component selects the corresponding node in the Parts tree. Double-clicking on the component replaces the list with its details.
END	No value	Displays a list of pipeline, cableway, and HVAC end parts. Clicking the component selects the corresponding node in the Parts tree. Double-clicking on the component replaces the list with its details.
OID	Type the OID value, including the curly-brackets ({OID value})	Selects the node for the component in the Parts tree.

For example, if you type ?COORD:29260.800 3962.400 5486.400 6 in the search box, the software displays the component with the corresponding COORD value in the tree, if one exists. When you click the listed component, the corresponding node in the Parts tree is selected. Double-click the listed component to display all of the component details.

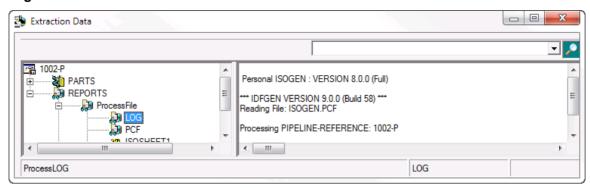
Parts

You can view the extracted parts by expanding the **Parts** node on the left side of the dialog box.

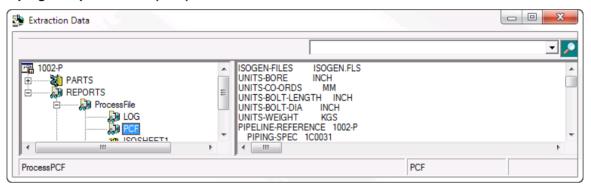


The **Reports** node includes:

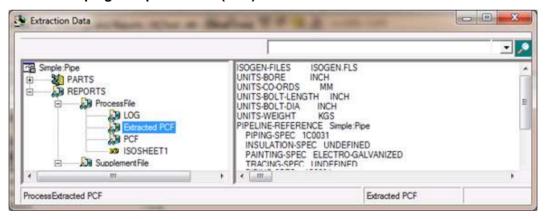
Log File



Piping Component File (PCF)

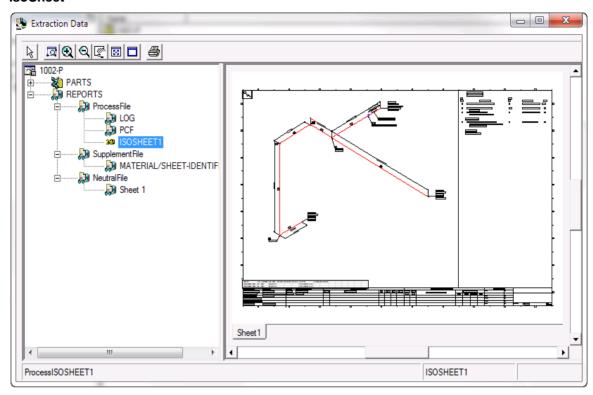


Extracted Piping Component File (PCF)



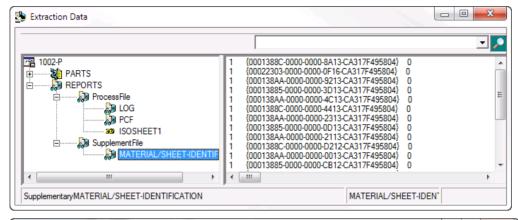
■ NOTE The extracted PCF is stored only if you have configured **Pre-processor**. For more information on how to store original extracted PCF, see *View Extraction Data* (on page 113).

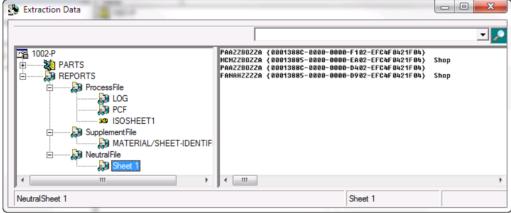
IsoSheet

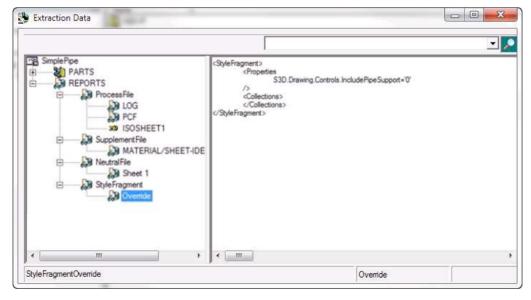


Other Report Files

Examples include the Supplement file, the Neutral file, and the Style Fragment file.







Create Piping Isometric Drawings by Query using Block Assemblies

The following procedure shows how to create a piping isometric drawing by query using block assemblies in the model. The block assemblies have the pipe parts grouped as needed to break down the design for construction requirements.

Set Up a Drawing

- ★ IMPORTANT You must be in the Drawings and Reports task to create a new folder and component.
- 1. Right-click the drawing root in the **Management Console**, and then select **New**.
 - The **Add Component** dialog box displays.
- 2. On the **General** Tab, select **Folder**, and then click **OK**.
- 3. In the **Management Console**, right-click the **New Folder**, and then select **Rename** to rename the folder as needed.
- 4. Right-click the renamed folder, and then select New.
 - The Add Component dialog box displays with the available drawing components.
- 5. Select **Isogen Isometric Drawings by Query** component on the **General** tab, and then click **OK**.
- 6. In the **Management Console**, right-click the isometric drawings component, and then select **Rename** to rename the folder as needed.

Define a Filter

- 1. To define a style for the drawing, right-click the piping assembly drawings component, and then select **Setup**.
- 2. Expand the Filter list, and then select More.

The **Select Filter** dialog box displays.

3. Click New Filter [₩].

The **New Filter Properties** dialog box displays.

- 4. In the **Name** box, type a name for the new filter.
- 5. Click the **Object Type** tab. In the hierarchy, expand **Planning**, and then select **Planning Hierarchy** > **Assembly**.
- 6. Click OK.
- 7. In the New Filter Properties dialog box, select the new filter, and then click OK.
- 8. In the **Setup** dialog box, click **OK**.
- TIP For more information on filters and filter properties, see *Select Filter Dialog Box* (on page 108) and Filter Properties Dialog Box.

Define a Style

1. To define a style for the drawing, right-click the piping assembly drawings component, and then select **Setup**.

The **Setup** dialog box displays.

2. Expand the Style list, and then select More.

The Select Drawing Style options window displays.

3. Select Iso Assembly style, and then click OK.

Create a New Assembly Iso Package

1. In the **Management Console**, right-click **Piping Assembly Drawings** component, and then select **Save Package**.

The Save Package dialog box displays.

- 2. Select the appropriate Package Name and Package Description.
- 3. In the Tab Name, select General.
- 4. Click **OK** to close the dialog box.
- **NOTE** You can save or delete the source assembly isometric drawing by query record.

Route Pipes

- 1. In the Piping task, route pipes as required. To route a pipe, see Create a new pipe run in the *Piping User's Guide*.
- 2. Use the commands on the **Route Pipe** ribbon bar to modify or route pipes. For more information on route options, see *Route Pipe* in the *Piping User's Guide*.
- NOTE You can skip this procedure if the model has pipelines already routed.

Generate Spools

- ★ IMPORTANT You must generate spools from the pipelines to create block assemblies for pipe runs. These block assemblies are used in individual isometric drawings.
- To create spools from the pipelines, see Create spools in the Piping User's Guide.

Create Blocks

- 1. In the Planning task, create blocks that split the pipelines. To create blocks, see *Split a block* in the *Planning User's Guide*.
- 2. Use the commands on the **Split Block** ribbon bar to modify and split the blocks. For more information on split blocks and split block commands, see *Split Block* in the *Planning User's Guide*.

Assign Pipe Parts to Blocks

1. To assign pipe parts to blocks, see *Start the block assignment process* in the *Planning User's Guide*.

■ NOTE To stop the block assignment process, see *Stop the block assignment process* in the *Planning User's Guide*.

2. After the process is complete, refresh the **Workspace**.

A new block is created in the **Assembly** tab. Expand the block and you can see that all the pipe parts are under the **UnAssigned Parts** node.

Assign Spools to Assembly Blocks

1. Click New (Block) Assembly .

The software prompts you to select an assembly or assembly block contents.

- 2. Select the appropriate spools within a block.
- 3. Type a name for the new assembly.
- 4. Click Accept .

The software prompts you to select the parent block.

- Select the parent block.
- 6. Click Finish.

The software creates a new assembly with spool(s) under the parent block.

7. Repeat the process as required.

Create Assembly Iso Drawing by Query

- 1. In the Drawings and Reports task, the right-click the folder you created when setting up the drawing, and then select **New**.
- 2. Select Drawings by Query Manager component, and rename it as needed.

The software creates a new component is created.

3. Right-click the new component, and then select **Setup**.

The software displays the **Setup** dialog box.

4. Under Filter select More.

The **Select Filter** dialog box displays.

Click New Filter[♥].

The New Filter Properties dialog box displays.

- 6. In the **Name** box, type a name for the new filter.
- 7. On the **Assembly** tab, select the assemblies to include in the filter
- 8. Click OK.

9. In the **Setup** dialog box, under **Package** select **More**.

The Select Package window displays.

- 10. Select the **Piping Assembly Iso** package, and then click **OK**.
- 11. Click **OK** to close the **Setup** dialog box.
- 12. In the **Management Console**, right-click the isometric piping assembly component, and then select **Run Query**.

The software creates the isometric piping assembly drawings under the **Drawings by Query Manager** folder.

13. Right-click the piping assembly drawing, and then select **Create Drawing(s)**.

The software creates isometric drawings for the assemblies created above.

14. Right-click an isometric drawing, and then select **Update Now**.

See Also

Isogen Isometric Drawings by Query (on page 99)

SECTION 8

Isometric Drawing Styles

Isometric drawing styles control several aspects of the isometric drawing output, including the output location and the type of object used for drawing creation. Each drawing style is also associated with a set of options and a backing sheet.

To customize the isometric drawing styles for your company, you can copy and then modify the delivered isometric drawing styles.

Delivered Isometric Drawing Styles

The delivered isometric drawing styles are Iso_Pipeline, Iso_Piperun, Iso_Spool, Iso_PenSpool, Iso_WBS, Iso_Stress, and Iso_System. Marine mode also delivers the Iso_Assembly drawing style. Each delivered style has an associated XML file and IGR file. The XML file contains the isometric options, and the IGR file is the backing sheet for the isometric drawing.

- **Iso_Pipeline** creates a final isometric drawing used to construct the model. This style creates one drawing per pipeline system. This drawing style is an example of a potential configuration for a fabrication isometric. It includes a material list.
- Iso_Spool creates an isometric drawing used in the fabrication shop to manufacture the pipe. This style uses piping spools. You can create spools using the Generate Spools command in the Piping task. Like the final isometric style, it includes a material list.
- Iso_WBS creates an isometric drawing that documents a collection of parts that are assigned to one Work Breakdown Structure (WBS) item of the type Group Iso Drawing. This style creates one drawing per WBS item.
- Iso_Stress creates a Piping Component File (PCF) that can be output to the CAESAR II pipe stress analysis software. No drawing is created. To save the PCF file, use the Save As command. For more information, see Retrieve piping component file data (PCF) (on page 74).
- Iso_PenSpool creates an isometric drawing that documents penetration spools that consist
 of a penetration plate and several piping spools. This style creates one drawing per
 penetration spools. You can create penetration spools using the Create Penetration
 Spools command in the Piping task.
- Iso_Piperun creates a draft of an isometric drawing for checking against project guidelines.
 This style creates one drawing per pipe run. This drawing style is an example of a drawing configuration used for checking pipeline designs prior to extracting the fabrication isometric.
- **Iso_System** creates an isometric drawing that documents a system, typically a piping system of connected pipelines. This drawing style creates one drawing for each system. Each pipeline change is called out on the drawing.
- Iso_Assembly (Marine mode only) creates an isometric drawing that documents piping block assemblies. This drawing style creates one drawing for each block. Assembly blocks split the pipelines into pipe parts (similar to spools). For more information, see Create Piping Isometric Drawings by Query using Block Assemblies (on page 121).

- Iso_HVAC creates an isometric drawing that documents duct systems within the model. A
 duct system organizes a collection of duct runs.
- Iso_CableTray creates an isometric drawing that documents cableway.

You can create other isometric drawing styles, such as a Bid style for construction contractors to bid on a project.

Migration of Isometric Drawing Styles

Migration of isometric drawing styles occurs automatically when you do one of the following two things:

- In migrated catalogs, the software only migrates the isometric drawing styles that already exist in the catalog (the XML style files that already exist in the catalog database as objects). For more information, contact Intergraph Support Services. You can find support information on our web site http://support.intergraph.com/).
- Automatically migrate your existing or modified isometric styles by opening them in ISOGEN
 Configuration r and saving them to the catalog. For more information, see *Edit Options*(Isogen Isometric Drawing Component Shortcut Menu) (on page 134).

Isometric Options

Isometric options define the isometric drawing output, which includes symbols, dimensions, layers, drawing frame attributes, material lists, weld lists, detail sketches, and many other settings. The options are stored in an XML file in the [Reference Data Product Folder]\SharedContent\PmfglsoStyleData folder on the server.

You can modify the option control data using the **Edit Options** command. This command opens **Isogen Configuration**, which is the tool used to control centrally all the options related to the appearance and information content of the styles of isometric drawings. If you directly manipulate the XML file, your changes may not take effect in the software.

You can also use the Edit Options command to import, export, and save symbol maps.

The **Option Override** command provides the flexibility of modifying option control data on a per drawing basis. This feature allows you to override style settings for a single Isogen isometric drawing without having to define a new drawing style.

The isometric option control data is integrated within the catalog reference data. You must have access rights to the catalog reference data to edit and save the option control data.

Backing Sheets

The backing sheet allows you to customize the drawing style with your company's logo, watermark, and drawing borders. The delivered backing sheets are stored as IGR files in the [Reference Data Product Folder]\SharedContent\PmfglsoStyleData folder on the server.

A document can include two different types of sheets: working sheets and background sheets. Working sheets contain design data, and background sheets contain title block graphics, borders, company logo, and watermarks. Each working sheet can contain a reference to a background sheet. If this reference is set, the size of the background sheet determines the size of the working sheet. Also, graphics on the background sheet become visible in the working sheet.

NOTE Graphic objects used in the background sheets must be embedded, not linked, using the **Insert > Object** command in **SmartSketch Drawing Editor** when editing the drawing or the drawing template.

When you create a backing sheet for isometric drawings, the backing sheet must have the following characteristics:

- It must contain a single, empty working sheet.
- It must contain a single background sheet containing the required graphics. The background sheet should be set to the appropriate size and scale.
- The working sheet must contain a reference to the background sheet.
- The working sheet must be active when you save the template.
- The working sheet should have layers set up with the required colors, unless the color is specified explicitly in the isometric options. If the working sheet does not have the required layers, Isogen creates the layers automatically and assigns them to the colors specified in the options.

Embedded Labels in Isometric Drawing Styles

To enable recursive expansion of embedded labels used in isometric drawings styles, the report RFM file must set the **ToParse** flag to **Yes**, as in the following example:

```
<DATA
Column="ShortMaterialDescription"
ToParse="yes"
Visible="yes"/>
```

See Also

Customize isometric style options (on page 171) Override isometric drawing style options (on page 196)

Isometric Drawing Types

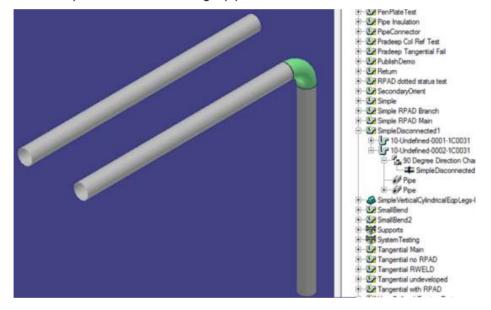
An isometric drawing usually consists of a number of parts that are physically connected so that they form a network such as a pipe run, pipeline or spool, or cableway. It is possible to create several other types of drawings using the Smart 3D > Drawing Creation > Overview > Drawing Class and the Smart 3D > Drawing Creation > Overview > Sort Input Parts style settings in Isogen Configuration.

NOTE You use **Isogen Configuration** to define the style settings that Isogen uses for isometric drawing generation and report output. For more information, see Isogen Configuration.

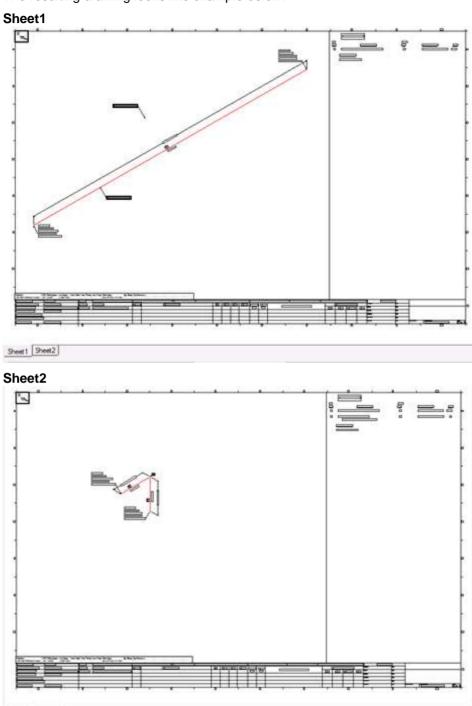
Disconnected Section Drawings

If the parts that make up a drawing consist of several networks of connected parts, you can set **Smart 3D > Drawing Creation > Overview > Sort Input Parts** to **Connected Sets** in the isometric style to enable support for multiple connected networks in the same drawing. Each network produces output as though it was in its own drawing but the output is collected into the same drawing file.

The example below shows a single pipeline that consists of two networks.



The resulting drawing looks like example below:



By default, each section is given a pipeline reference as normal. If any duplicates are created, then a number is appended to make the references unique.

Multi-Pipeline Drawings

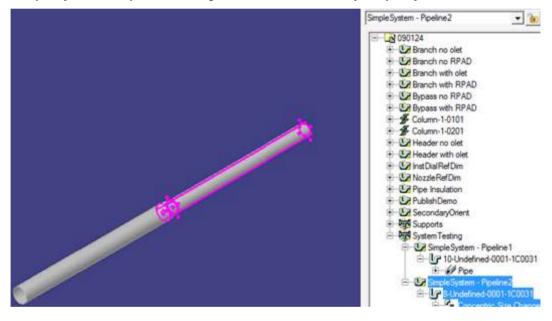
If the parts that make up a drawing consist of more than one pipeline, you can set **Smart 3D > Drawing Creation > Overview > Sort Input Part** to **Pipeline** in the isometric style to enable support for a multi-pipeline drawing. Each pipeline outputs in a single drawing file as a disconnected section with its own pipeline reference. This is useful for groups of related pipelines that are not connected, such as the equipment trim.

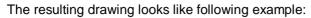
System Drawings

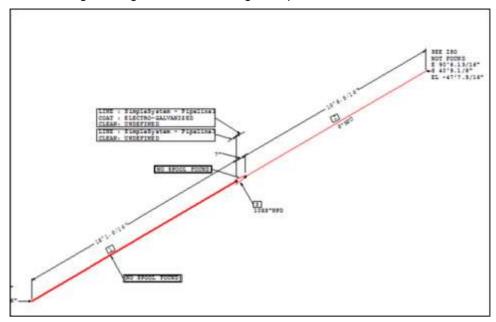
If the parts that generate the drawing consist of more than one pipeline and they are connected as a single network, you can set **Smart 3D > Drawing Creation > Overview > Drawing Class** to **System** in the isometric style to create an Isogen system drawing. Any boundaries between the pipelines in the system are indicated on the drawing.

▶ NOTE If Smart 3D > Drawing Creation > Overview > Drawing Class is set to System, the software ignores the Sort Input Parts setting.

The example below shows a drawing of the following pipelines **SimpleSystem – Pipeline 1** and **SimpleSystem – Pipeline 2** using a WBS Item called **VerySimpleSystem**.







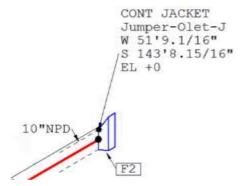
Jacketed Pipeline Drawings

If the parts of a drawing consist of jacketed pipelines, the jacket pipe is drawn as a hollow pipe to indicate the presence of the core pipe, and each connection between the jacket and core pipes are indicated with continuation references. The core pipes display continuation references for jacket pipes whereas jacket pipes display continuation references for core pipes.

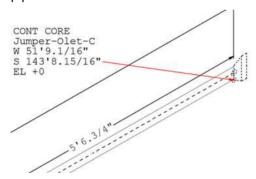
To generate jacketed pipeline diagrams, make sure that the jacket and core pipe association has been established. For more information, see *Placing the Jacket Pipe and Components* in *Piping User's Guide*.

NOTE You must use the JFWN (weld neck jacket flange) and JFSO (slip on jacket flange) three-port flanges SKEYs for jacket flanges. For more information, see *Flanges* (*SKEYs*) (on page 652).

The example below shows a core pipeline drawing with a continuation reference for the jacket pipe.



The example below shows a jacket pipeline drawing with a continuation reference for the core pipe.



Isometric Style Common Tasks

The following tasks are used when you create new isometric styles.

Modify Drawing Border Files

You can create or modify border files two ways. You can modify existing delivered border files. For more information, see *Modify an Existing Border File* (on page 489).

If you have existing MicroStation J (V7) DGN files you want to use as a drawing border, you can import them. For more information, see *Import an Existing MicroStation DGN Border* in the *Drawings and Reports Reference Data Guide*.

Create New Isometric Styles

You can add new isometric styles to the delivered BulkLoadIsoKeys.xls file, and then bulkload the changes into the model data. For more information, see *Create a New Isometric Drawing Style* in the *Drawings and Reports Reference Data Guide*.

Use Isogen Configuration

You can modify, save, and import style options using **Isogen Configuration**. For more information, see *Customize isometric style options* (on page 171).

Importing Data from the Style XML File

You can import style data from the style XML file directly through **Isogen Configuration**. When you import styles, they are shown immediately within ISOGEN Configuration. For more information, see Import data from the style XML file.

Develop the Look and Feel of Drawings

You can change the look and feel of an isometric drawing by changing options within **Isogen Configuration**. Options can specify everything from drawing content to the system controls for output definition. You can also use the **Drawing Setup Tool** to customize the content and format of the isometric drawing for a specific drawing style. You can customize the drawing frame and specify which reports are plotted on the isometric drawing when it is generated. You can also define the layout and appearance of the plotted report data. For more information, see *Drawing Setup Tool* (on page 234).

Override Drawing Style Options

You can override isometric drawing style options on a per drawing basis. For more information, see *Option Override (Isogen Isometric Drawing Shortcut Menu)* (on page 196).

Use Alternative Text

Alternative Text, or AText, is an Isogen feature that allows you to change or remove any text on the isometric drawing. You can substitute your own text in the place of standard Isogen words. For more information, see Alternative Text Options.

Map Isometric Data to Drawing Layers

You can map layers within a previously created drawing border file to isometric data. The options used to define the mapping are found in **Isogen Configuration** in the **Drawing Area** >

Graphics > Definitions and **Drawing Area > Graphics > Layers** style groups. For more information, see *Map isometric data to drawing layers* (on page 215).

Configure the Material List

You can specify three different styles for material lists on isometric drawings. You use **Isogen Configuration** to set the options for the material list.

Assign Labels

You assign labels to attributes within a drawing through **Isogen Configuration**. For more information, see *Assign labels* (on page 184).

Populate the Title Block

You can use labels to customize the title block of a drawing. Labels are often used for single pieces of data, such as the approval date or your company name. You can use **Isogen Configuration** to specify options for the appearance and content of the title block. For more information, see *Populate the title block* (on page 185).

Setting the Symbol Mapping

You can set symbol mapping using **Isogen Configuration**. For more information, see *Symbol Map* in the Smart 3D *Isometric Drawing Options Reference Guide*, which is available using the **Help > Printable Guides** command in the software.

Edit Options (Isogen Isometric Drawing Component Shortcut Menu)

Opens **Isogen Configuration** so you can set options for isometric drawing output, such as dimension styles, layers, drawing frame text, material lists, and weld lists.

★ IMPORTANT Some of the workflows described in this section are not supported by all Isogen isometric drawing types. For example, a workflow that is applicable to a piping isometric drawing may not be applicable to cabletray and HVAC isometric drawings.

You can also use the **Edit Options** command to import, export, and save symbol maps. You can import isometric styles from a previously created and saved XML file containing isometric keys.

You can access this command by right-clicking an Isogen Isometric Drawings component or package in the **Console**. You must have access rights to the catalog reference data to save isometric options to the Catalog database.

The **Edit Options** command and **Isogen Configuration** enable you to set many isometric drawing style options:

Configure Drawing and Report Settings (on page 170)

Fonts in Isometric Drawings (on page 198)

Flow Arrows (on page 202)

Bending Files (on page 204)

Material Lists and Material List Files (on page 205)

Pipeline Lists (on page 211)

Weld Lists and Weld Files (on page 212)

Change Management in Piping Isometric Drawings (on page 222)

■ NOTE For more information on isometric drawing style options, see the Smart 3D *Isometric Drawing Options Reference Guide*, which is available with the **Help > Printable Guides** command in the software.

SECTION 9

Isogen Configuration

Sets options for the active isometric drawing style. To access **Isogen Configuration**, right-click an Isogen Isometric component or package in the **Management Console** or **Drawing Console**, and then select **Edit Options** on the shortcut menu.

In Isogen, the isometric drawing style controls all of the options related to the drawing and report appearance and information content produced during isometric drawing generation. Each isometric drawing style contains a group of object properties that control some aspect of the drawings and reports that are produced by 2D piping solutions and 3D plant design software products. These properties are stored in a style file. During isometric drawing generation, the contents of the style file are converted into the set of text control files that are read by Isogen. As such, the way that you configure a style determines what the isometric drawing looks like when the drawing is generated, as well as what information is included in a report. Multiple drawing styles can exist side by side to produce different outputs from the same design file.

★ IMPORTANT In this version of the software, cable tray and HVAC isometric drawing options are limited to beta access. As such, isometric options specific to cable tray and HVAC drawings are not visible in **Isogen Configuration** by default. To turn on cable tray and HVAC options so that you have access to the full array of isometric drawing settings provided by the software, please contact *Intergraph Support* (http://www.intergraph.com/support).

Isogen Configuration provides a single, streamlined interface for setting isometric drawing style properties. The **Isogen Configuration** environment provides easy access to Isogen's substantial array of controls and files and enables you to configure projects quickly and efficiently. Any modifications that you make can then be saved to the drawing style file for future use.

Isogen Configuration also includes the tools listed below. When used, these tools can further streamline the process for configuring drawing output.

- Drawing Setup Tool includes a limited set of drawing options that can assist you in configuring the drawing template. The drawing template is usually a company standard backing sheet on which Isogen plots the drawing and related data, such as the material take-off, welding report, and project-related data. The Drawing Setup Tool has a graphical interface, overlaying the location of key items onto the backing sheet. For more information, see Drawing Setup Tool (on page 234).
- Detail Sketch Manager helps you configure detail sketches for isometric drawing output. The Detail Sketch Manager allows you to place parameters on the selected detail sketch so that the correct relevant values, such as weld numbers, part numbers, and angles, are plotted on the isometric drawing. For more information, see Detail Sketch Manager (on page 229).

Key features of Isogen Configuration include:

- Access to all of the Isogen controls using a streamlined user interface.
- Configure the drawing template and detail sketches for isometric drawing output using convenient wizards.
- Save style changes to an XML file.
- Review the impact spontaneous style modifications have on drawing output using the built-in drawing previewer.
- Locate a specific property based on user-specified criteria using the Search feature.
 Similarly, you can use the Search feature to locate configuration options for a specific Isogen report.

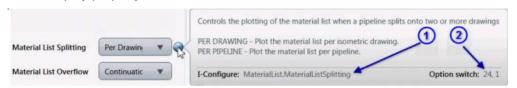
See Also

Isometric Drawing Styles (on page 125)

Get online Help information

As you work, you can use the resources listed below for assistance using **Isogen Configuration**:

- Online Help opens the standard online Help system that comes with the software. To access online Help, click and then select ISOGEN Configuration Help on the Help sub-menu. Alternatively, click Help? on the far right-side of the Isogen Configuration title bar.
- Context-sensitive Help provides immediate information about isometric drawing style property settings. When a property is in focus in the Options panel, press F1 to display the corresponding online Help topic
- Tool Information displays details about each property, including the corresponding I-Configure property name (1) and the option switch that the property defines (2). In many instances, details about the property settings are available. Mouse over Tool Information to display property details.



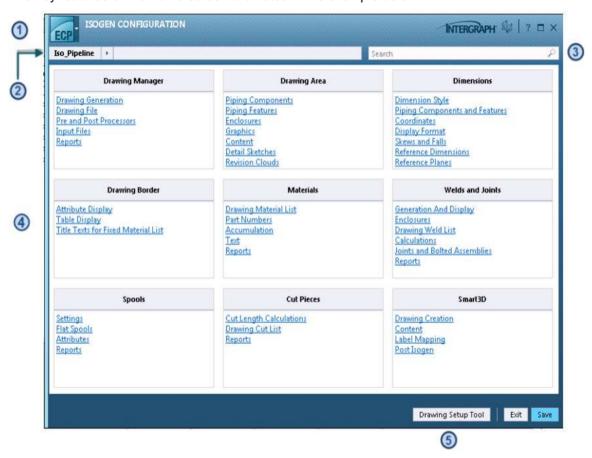
- TIP In instances where I-Configure or option switch information is not applicable, the software displays **N/A**.
- Printable Guides opens the online Help in a printable guide format. Click select Help > Printable Guides to access any of the following documents provided in PDF format.
 - Isometric Drawing Options Reference Guide
 - Isogen Symbol Key (SKEY) Definitions Reference Guide
 - Isogen Isometric Drawings User's Guide
 - **NOTE** You need Adobe Acrobat version 9.x or later to open the PDF files.

Isogen Configuration Interface

Each time that you open **Isogen Configuration**, the Home screen displays. The Home screen provides the work area for selecting the isometric drawing style category that contains the properties to configure, accessing the **Drawing Setup Tool**, searching for specific isometric drawing properties to view and edit or Isogen report options to configure.

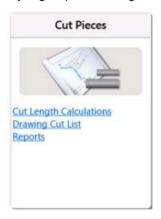
NOTE From the Home screen you can open the **Configuration** view, which provides direct access to all of the isometric drawing style properties as well as properties that control and define report output. For more information, see *Configuration View* (on page 143).

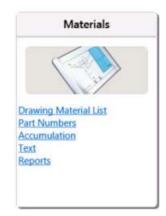
The key features of the Home screen are noted in the example below.



- 1. The **Menu** provides access to setting drawing units, viewing user documentation, importing and exporting style data, and updating the style file with any changes. For more information, see *Isogen Configuration Menu* (on page 141).
- 2. The **Navigation** bar displays the **Isogen Configuration** breadcrumb trail, which indicates the active isometric drawing style, and, when applicable, the drawing style category and style group that are currently selected. For more information about navigating the software, see *Navigation Techniques* (on page 155).

- 3. The Search box allows you to locate isometric drawing style properties based on a keyword, option switch, property name, or AText number search. You can also search for configuration options related to a specific Isogen report, such as the Weld Summary report or the Material Control File. For more information, see Search Filters (on page 160).
- 4. Style category panes. Each style category is broken down into a collection of related style groups. For example, the Cut Pieces category includes three style groups: Cut Length Calculations, Drawing Cut List, and Reports, whereas the Materials category includes five style groups: Drawing Material List, Part Numbers, Accumulation, Text, and Reports.





Each style group contains a subset of related grouping of Isogen isometric drawing controls. For detailed information about each style category and its respective isometric drawing properties, refer to the *Isometric Drawing Options Reference Guide*. You can access the guide using the **Help > Printable Guides** command in the software.

Isogen Configuration for Smart 3D contains the nine style categories listed below.

- Drawing Manager sets options for system controls, including pre- and post-processors and the names of the input files used by Isogen. The Drawing Manager category also supports the definition of all of the Isogen reports.
- Drawing Area controls the content that displays on the isometric drawing. The Drawing
 Area category also includes properties that define pipeline component representation on
 the drawing.
- Dimensions controls the display and format of dimensions and coordinate data on the isometric drawing.
- Drawing Border specifies the attributes plotted on the drawing frame, as well as their placement within the drawing frame. The Drawing Border category also includes properties that define symbol shape positioning.
- Materials controls the position, format, and contents of the parts list.
- Welds and Joints controls the display of welds on the isometric drawing. An option is available for defining the method that Isogen uses to calculate weld diameter totals. The Welds and Joints category also includes properties that set options for weld-specific reports. The weld list is the report that appears in the drawing, while the weld list summary contains the weld data formatted in a text file.
- Spools defines spool attribute data that is output on the isometric drawing. The Spools
 category also includes properties that define spool attribute data to summary report
 files, such as the spool information file.

- Cut Pieces sets properties for the cut list, which provides a list of pipes that are cut into smaller lengths during construction. You can also define the layout of the fixed and user-defined cut list formats and set options for the cut list summary file. Properties are also available for controlling cut length calculations.
- Smart 3D sets Smart 3D-specific options that define drawing output, which includes drawing symbols, dimensions, layers, drawing frame, attributes, and detail sketches.
- **NOTE** When you select a style group in one of the category panes, the software opens the **Configuration** view, which displays an additional subset of options. For more information, see *Configuration View* (on page 143).
- 5. Drawing Setup Tool opens the Drawing Setup Tool, which you can use to quickly configure the drawing template. The drawing template is usually a company standard backing sheet on which Isogen plots the drawing and related data, such as the material take-off, welding report, and project-related data. The setup tool has a graphical interface, overlaying the location of key items onto the backing sheet. For more information, see *Drawing Setup Tool* (on page 234).

Isogen Configuration Menu

Displays commands for setting drawing units, accessing user documentation, and updating the style file with any changes. The commands listed below are available on the menu:

Home

Returns you to the Home screen.

Options

Opens the **Options** dialog box so you can specify the drawing units at the application level. For more information, see *Set drawing units* (on page 181).

Save

Writes the current property settings and parameters to the active style file. You can also save property settings by clicking at the bottom of the Home screen.

Export Style Fragment

Opens the **Export Style Fragment** dialog box so that you can specify export settings and select the isometric drawing style properties to export. For more information, see *Export an isometric drawing style fragment* (on page 176).

Import Style Fragment

Opens the **Import Style Fragment** dialog box so that you can select a style fragment to import. For more information, see *Import data from a style XML file* (on page 173).

Help

Allows you to access product information. For more information, see *Get online Help information* (on page 137).

Isogen Configuration Help opens the online Help. You can also access the online
 Help by clicking Help? on the Home screen title bar.



- Printable Guides opens the delivered PDF manuals. You need Adobe Acrobat version 9.x or later to open PDF files
- Technical User Forum opens the Intergraph Technical User Forum website. This site contains user discussions on using Intergraph products.
- About ISOGEN Configuration provides Isogen Configuration version information.

Exit Application

Closes **Isogen Configuration**. You can also close **Isogen Configuration** by clicking at the bottom of the Home screen.

Status and Error Messaging

As you work, **Isogen Configuration** can display status and warning messages at the top of the **Preview** window in the *Configuration view* (on page 143). In addition to the text description, each message type has an associated color code.

• **Error** represents an error message for which an operation had to be stopped, as shown in the example below:

ISOGENControlPanel has encountered a problem and needs to close.

Warning represents a warning message for which an operation could not be completed, as shown in the example below:

A The path to the Detail Sketches directory e\SampleIsoDir\Piping_Project2\Data\Sketches\ does not exist

Status represents a general status message.



★ IMPORTANT Click X on the right side of the message box to close the message. Otherwise, the message continues to display. As further warning or status messages display, they accumulate in the messaging section with the most recent message displaying at the top.

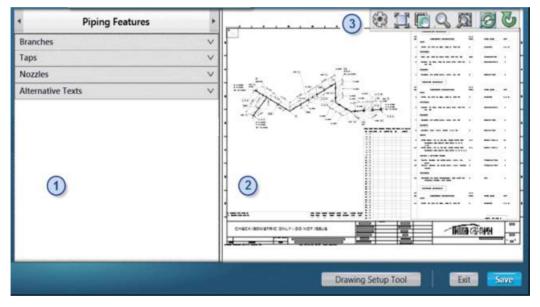
SECTION 11

Configuration View

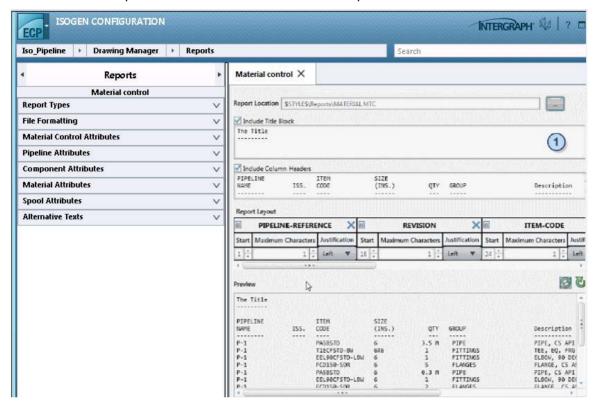
You use the **Configuration** view to control all the properties related to the appearance and information content of the active isometric drawing style. You also use this view to define the output location, content, and layout of the Isogen reports. The software automatically displays the **Configuration** view whenever you select a style group on the Home screen.

The **Configuration** view window is divided into left and right panels. The **Options** panel is on the left side of the window (1). The panel on the right side depends on the type of Isogen controls that you are configuring. For example, if you are configuring drawing output, the **Configuration** view shows the **Drawing Preview** panel (2). As you configure the active style, the drawing dynamically updates so that you can view the effect of your drawing property settings.

NOTE The **Drawing Preview** panel also includes a toolbar (3). You can use the commands on the toolbar to control the displays of data in the panel. For more information, see *Control the display of data* in *Drawing Preview Panel* (on page 146).



When you define report output, the **Configuration** view shows the **Report Definition** panel (1). The **Report Definition** panel simplifies the process for configuring Isogen reports. The example below shows the report definition for the Material control report.



Options Panel (on page 145)

Drawing Preview Panel (on page 146)

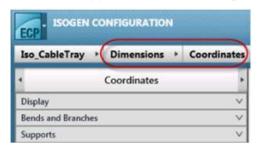
Report Definition View (on page 150)

What do you want to do?

- Fit all drawing objects (on page 148)
- Pan the view (on page 148)
- Zoom in on an area or zoom out from a point (on page 148)
- Magnify an area of the preview drawing (on page 148)

Options Panel

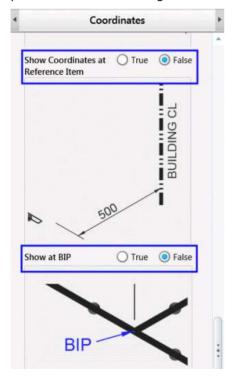
Organizes the groups of options that belong to the active style group. The example below shows the four options groups that belong to **Dimensions > Coordinates**: Display, Bends and Branches, Supports, and Heat Tracing.



NOTE The name of the active style group also displays as the title of the **Options** panel. The example above shows the **Coordinates** panel.

To expand an options group and display all of its related isometric drawing style properties, click the \sim icon that is next to the options group heading.

You can modify any of these properties as needed. Some properties have a selection list from which you can select a value. Other properties require you to type a value or text string. Changing the value of each property ultimately determines the appearance of the isometric drawing and report file content produced during drawing generation. In the example below, changing the **True/False** settings control whether coordinates for reference items and break-in points are shown on the generated drawing.

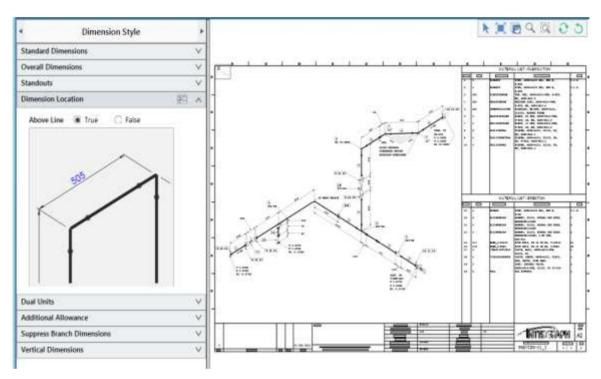


NOTE Click to collapse the options group and hide its properties.

Drawing Preview Panel

Displays a sample isometric drawing that is representative of the current isometric drawing property settings. The preview drawing is based on a sample piping component file (PCF) delivered with the software.

★ IMPORTANT The drawing preview does not reflect the drawing options that you have set in the Smart 3D category.



As you configure drawing properties, the drawing view updates so you can see the impact a property setting has on the drawing output. The examples below show the effects of changing the **Above Line** setting from **True** (1), where dimensions are placed above the dimension line, to **False** (2), where dimensions are inserted into the dimension line.



NOTE By default, the drawing view in the **Drawing Preview** panel updates automatically whenever a drawing property is modified. If **Automatically Refresh** is turned off, you must click **Refresh** on the **View** toolbar to manually update the drawing view.

Control the display of data

The **View** toolbar provides commands for manipulating the view of the example drawing in the **Drawing Preview** panel. For example, you can adjust the view to display the entire drawing, or you can focus on specific parts of the drawing. You can also increase or decrease the viewing area. Decreasing causes everything within the **Drawing Preview** panel to appear larger, while increasing causes everything within the window to appear smaller. The commands listed below are available on the **View** toolbar:

- **Select** selects a specific object in the isometric drawing. When you select an object in the preview drawing, it highlights in red.
- **Fit View** displays the entire contents of the isometric drawing in the **Drawing Preview** panel. For more information, see *Fit all drawing objects* (on page 148).
- **Pan** repositions the drawing in the **Drawing Preview** panel so that you can view another section of the drawing without changing the view magnification. For more information, see *Pan the view* (on page 148).
- Zoom increases or decreases the display size of the drawing in the **Drawing Preview** panel. You can zoom in to get a closer view of an object or zoom out to view more of the drawing at a reduced size. For more information, see *Zoom in on an area or zoom out from a point* (on page 148).
- **Zoom Area** increases the view magnification of an area in the isometric drawing that you define by drawing a fence around the part of the drawing to enlarge. For more information, see *Magnify an area of the preview drawing* (on page 148).
- Automatic Refresh controls whether the software automatically updates the view of the drawing in the **Drawing Preview** panel after a setting is changed. By default, this command is active. When the command is turned off, you must use **Refresh** to manually update the isometric drawing preview.
- Refresh updates the view of the drawing in the **Drawing Preview** panel to display any changes to the isometric drawing settings that you have defined since the last refresh action.

NOTE To cancel the active **View** command, click **Select**, or right-click anywhere in the **Drawing Preview** panel.

What do you want to do?

- Fit all drawing objects (on page 148)
- Pan the view (on page 148)
- Zoom in on an area or zoom out from a point (on page 148)
- Magnify an area of the preview drawing (on page 148)

Fit all drawing objects

Click **Fit View** on the **View** toolbar.

The display in the **Drawing Preview** panel updates so that the entire drawing is visible.

Pan the view

1. Click Pan e on the View toolbar.

The cursor changes to



2. Click to select a point in the **Drawing Preview** panel, and then drag the cursor up, down, left, or right to view other areas of the drawing.

Zoom in on an area or zoom out from a point

1. Click **Zoom** On the **View** toolbar.

The cursor changes to \mathbb{Q} .

- 2. Perform the following actions to zoom in and out:
 - To zoom in, click a starting point in the **Drawing Preview** panel, and then drag the cursor upward in the panel to increase the view size of the drawing area.
 - To zoom out, click a starting point in the **Drawing Preview** panel, and then drag the cursor downward to reduce the view of the drawing area.

■ NOTES

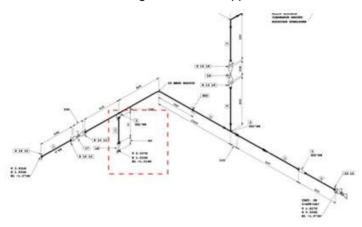
- Click **Fit View** to fit the entire drawing in the **Drawing Preview** panel.
- When you roll the IntelliMouse wheel backward when the Zoom command is active, the drawing view zooms out at the current cursor location. When you roll the wheel of the IntelliMouse forward, the drawing view zooms in.

Magnify an area of the preview drawing

1. Click **Zoom Area** on the **View** toolbar.

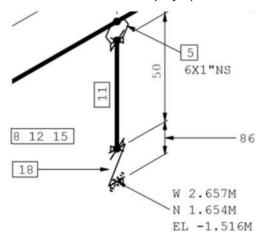
The cursor changes to +

2. Click a start point in the **Preview** window and drag to an end point in the active view to fence the area to enlarge. The fence appears as a red dotted line.



3. Release the mouse.

The Preview window display updates to show the selected area.



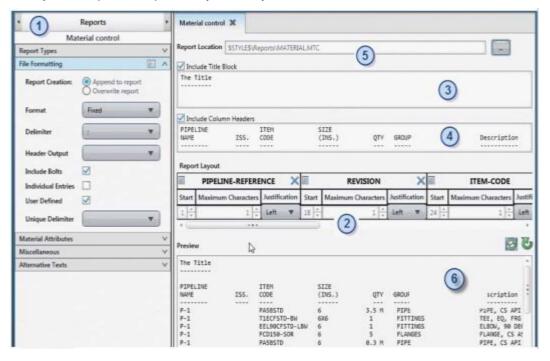
TIP Click Fit View to display the entire drawing in the Preview window again.

SECTION 12

Report Definition View

Drawing Manager > Reports organizes all of the reports that Isogen is able to output during isometric drawing generation. In addition to this global **Reports** panel, the Materials, Welds, Spools, and Cut Pieces categories each contain a content-specific subset of report types. For example, you can use **Welds > Reports** to define the weld summary and site weld information reports, while **Spools > Reports** enables you to define the spool information report. For more information about the global **Reports** panel, see Reports.

Whenever the **Reports** style group is active (1), the **Configuration** view switches to the **Report Definition** view. The report options that display in this view depend on the type of report being defined. Some reports provide options for defining the report layout, including which attributes are output (2) and whether the title block (3) and column headers (4) display. All report types require that you specify an output location and name for the report file (5). For many report types, the software displays a preview of the report at the bottom of the panel (6). As you modify the layout, the preview updates dynamically.



Dynamic File Naming

You can pre-process the Isogen.fls file to substitute a macro for one of the piping component file (PCF) attributes taken from the current input file. For example, setting **Report Location** to **\$Style\$\\$PIPELINE-REFERENCE\$.txt** outputs the selected Isogen report to the active style folder and names it using the PIPELINE-REFERENCE, as defined in the input PCF.



NOTES

- The Isogen.fls file appears in the first line of the PCF header. This file specifies all of the input and output filenames that Isogen uses during the isometric drawing generation process.
- If the required attribute is not present in the PCF, Isogen uses the NAME attribute instead.
- You can use system environment variables, such as TEMP, USERNAME, or COMPUTERNAME, in folder or file paths for report files.

The PCF attributes that are supported are listed in the table below.

PCF Attribute	Description
AREA / BATCH	Specifies a section of the plant or project. These two attributes are used interchangeably in the PCF.
ATTRIBUTE00 <i>to</i> ATTRIBUTE199	Specifies the user-defined general purpose pipeline attributes. These attributes can be defined and used for any purpose.
CLIENT-DRAWING-IDENTIFIER	Specifies the user-defined string used to identify, track, or control each individual drawing without using the usual pipeline reference or sheet number identification. Typically, this attribute is output on spool isometric drawings.
DATE-DMY	Specifies the date associated with the pipeline.
EQUIPMENT-REFERENCE	Specifies the name of the equipment. This attribute is mandatory when defining an equipment item.
INSULATION-SPEC	Specifies the default insulation specification.
JACKET-SPEC	Identifies the default piping specification for the components in the pipeline that form part of the jacket.
MISC-SPEC1 to MISC-SPEC5	Indicates the user-defined specification references that hold arbitrary data.

PCF Attribute	Description
NOMINAL-CLASS	Specifies the pipeline design class.
NOMINAL-RATING	Specifies the pressure rating of the pipeline.
OUTPUT-FILE-NAME	Specifies the filename of the isometric drawing output plotfile.
PAINTING-SPEC	Identifies the default painting specification.
PCF-FILE-NAME	Specifies the name of the input piping component file.
PIPING-SPEC	Identifies the primary specification to which the pipeline belongs.
PIPELINE-DRAWING-SEQUENCE -NUMBER	Specifies the user-defined text string used to identify, track, or control each individual drawing without using the pipeline reference.
PIPELINE-REFERENCE	Specifies the line reference or ID of the pipeline.
PIPELINE-TEMP	Specifies the pipeline operating temperature.
PIPELINE-TYPE	Indicates the type of pipeline construction.
PROJECT-IDENTIFIER	Specifies the project number or name.
REVISION	Indicates the revision identifier for the pipeline.
SPOOL-DRAWING-SEQUENCE -NUMBER	Specifies the spool sheet isometric drawing sequence number.
SPOOL-NAME / SPOOL- IDENTIFIER / SPOOL-ID	Specifies the name of the spool. These three attributes are used interchangeably in the PCF.
SPOOL-PREFIX	Specifies the string used when creating spool identifiers.
SYSTEM-ISOMETRIC-REFERENCE	Specifies the name of the drawing generated from a system piping object data (POD) file.

Report Definition Options

★ IMPORTANT The options listed below are not available for all report types. For example, some reports allow you to only define the report output location and filename. Other reports allow you to specify attribute data and define the layout.

Report Location

Specifies the output location and name of the report. Type the full path and filename of the report. Alternatively, click **Browse**, and navigate to the output location. Setting this option is mandatory.

■ NOTE The filename itself can be dynamic. For example, you can use the pipeline reference as part of the filename. For more information, see *Dynamic File Naming* in *Report Definition View* (on page 150).

Include Title Block

Controls the output of a report title. To include a title, select the checkbox.

Title Block

Specifies the report title output by Isogen. Type the title of the report in the text box. Use the SPACEBAR and ENTER keys to insert spaces and blank lines. This option displays only if **Include Title Block** is selected.

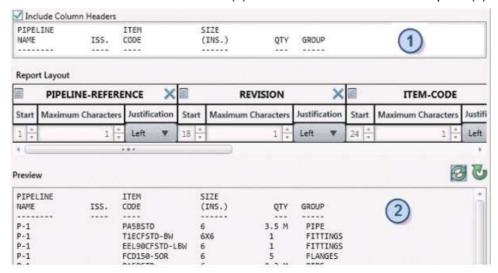
Include Column Headers

Controls the output of column headers. To include headers, select the checkbox.

Column Headers

Specifies the display name for each column of attribute data that is output in the report. This option displays only if **Include Column Headers** is selected.

In the text box, type the display name for each column that you have selected for output. Use the SPACEBAR to insert empty spaces between each display name. To insert a blank row so that you can type another line of text, press ENTER. The display in the **Preview** pane updates as you enter text. The example below shows the relationship between the content of the **Column Headers** box (1) and the content in the **Preview** pane (2).



Report Layout

Displays the attributes selected for the active report. Each attribute translates to a column of data in the report. For each attribute, you can define the data listed below:

- Comments opens a text box so that you can type a text comment for any definition contained in the previous column. If any text is entered in this field, the text is output as a comment--preceded by a !--to the relevant data or summary report file.
- Start specifies the starting point for the column.
- Maximum Characters defines the maximum number of characters allowed in the column.
- Justification defines the justification of the text within the column. Select Left, Right, or Numeric.
- **Enabled** controls the display of data in the report file for the component group defined by the **Group** setting. Select the checkbox to display component data for the specified group. This option displays only for the neutral report.
- Group specifies the component group output in the current report file record. Select a component group in the list. This option displays only for the neutral report.

NOTE To add an attribute to the report layout, open an options group and double-click the attribute name. For more information, see *Define the report layout* (on page 206).

Preview

Displays a preview of the report content based on the current report definition parameters. When you first begin defining a report, a **Preview (file not found)!** error message displays until you specify the output location in the **Report Location** box.



Automatic Refresh

Controls whether the software automatically updates the report **Preview**. By default, this command is active. When the command is turned off, you must use Refresh to manually update the isometric drawing preview.



Refresh

Updates the view of the report to display any changes to the layout that you have made since the last refresh action.

Navigation Techniques

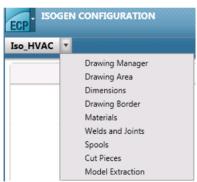
Isogen Configuration organizes the hundreds of Isogen drawing controls using a hierarchical structure that consists of three levels of content: style category, style group, and option group.

NOTE Each option group is further broken down into collections of related drawing properties. For more information, see *Options Panel* (on page 145).

This hierarchical structure breaks down the wide array of Isogen controls into manageable chunks of information. To help you keep track and know where you are within the hierarchy, **Isogen Configuration** uses a location-based breadcrumb navigation style. By providing a breadcrumb trail that illustrates how the hierarchy of Isogen drawing controls is structured, you can use one-click access to freely move through the hierarchy to configure drawing properties.

The breadcrumb trail indicates in which collection of drawing properties you are, and more importantly allows you to navigate your way through the product to find other collections of drawing properties. The breadcrumb trail starts with the name of the active style. After this, the active style category is listed, followed by the active style group.

In the example below, the **Iso_HVAC** breadcrumb represents the active style and provides links to its related style categories.



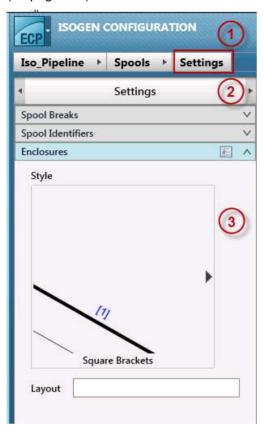
NOTE Each style category is synonymous to a category pane on the Home screen. For more information, see Review the Interface.

In the next example, the **Spools** breadcrumb represents the active style category and provides links to its collection of style groups.



The **Settings** breadcrumb, shown in the example below, represents the active style group (1). Each style group contains a collection of related option groups. When a style group is active, each options group that it contains is listed in the **Options** panel (2).

NOTE Each options group contains a collection of related drawing properties (3), which are used to configure isometric drawing and report output. For more information, see *Options Panel* (on page 145).

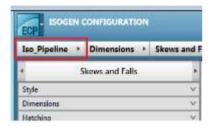


What do you want to do?

- Return to the Home screen (on page 157)
- Switch to a different style category (on page 157)
- Switch to a different style group in the active category (on page 158)
- Navigate the Options panel (on page 159)

Return to the Home screen

• Click the active style breadcrumb in the **Navigation** bar to return to the Home screen.



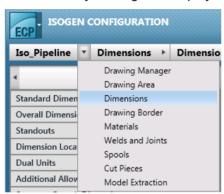
Alternatively, click and then select **Home**.

Switch to a different style category

1. In the **Navigation** bar, click ▶ on the active style breadcrumb.



A list of the style categories displays.



2. Select the style category. For example, to access those properties specific to the material list, select **Materials**.

The software switches to the **Materials** category (1). By default, the name of the first style group in the Materials hierarchy is shown on the style group breadcrumb and appears as the title of the **Options** panel (2). A list of the respective options groups is also displayed in the panel.



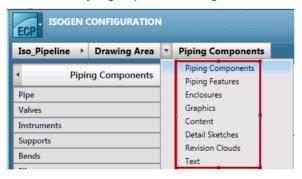
■ NOTE Alternatively, you can *return to the Home screen* (on page 157), and then navigate to the appropriate style category and style group.

Switch to a different style group in the active category

1. In the **Navigation** bar, click ▶ the active style category breadcrumb.

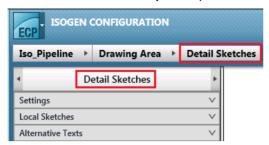


A list of the style groups that belong to the active style category displays.



2. Select a style group. For example, to access properties related to detail sketches, select **Detail Sketches**.

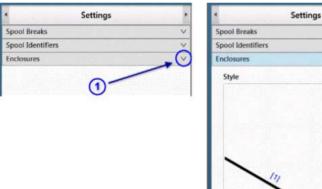
In the example below, Detail Sketches is now the active style group, as indicated on the breadcrumb and in the **Options** panel title bar.

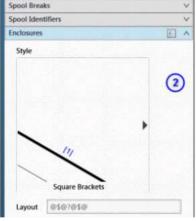


NOTE Use **Forward** and **Back** on the **Options** panel title bar to move sequentially through the style groups that belong to the active style category. Alternatively, you can *return to the Home screen* (on page 157), and then select the style group in the style category pane.

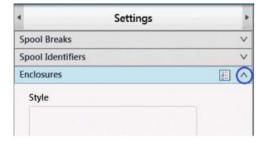
Navigate the Options panel

■ Use **Expand** ∨ and **Collapse** ∧ to display and hide the properties within an options group. In the example below, clicking **Expand** ∨ on the **Enclosures** label **(1)** displays the collection of properties in **Spools** > **Settings** > **Enclosures** (2).





■ NOTE To hide the properties, click **Collapse** ∧.



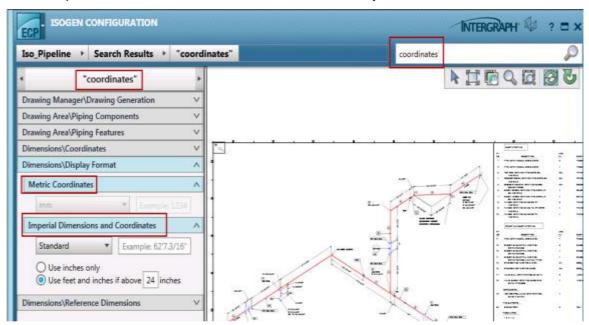
SECTION 14

Search Filters

Because of the hundreds of isometric drawing properties that are available, locating a specific one can be a daunting task. The **Search** feature lets you search for a specific options group or individual drawing property based on the following user-specified criteria: a keyword, an option switch, an I-Configure property name, or an AText number. As searches are completed, the software maintains a search history, which allows you to locate and re-use search criteria quickly and easily.

Keyword Search

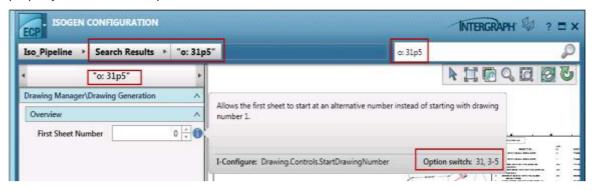
In a keyword search, the software locates all of the options groups or drawing properties that contain the keyword, or text string, that you specify in the **Search** box. In the Options panel, the software displays a list of every style category and style group in which the keyword appears. The example below shows the search results based on the keyword **coordinates**:



Option Switch Search

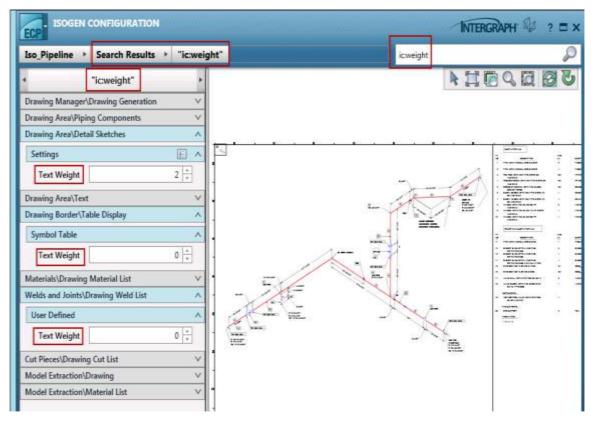
In an option switch search, the software locates those properties that define the specified option switch. You can direct the software to search only for the option switch number or, for option switches that have multiple settings, search for the option switch and the switch position. As with the keyword search, the results are displayed in the **Options** panel. For example, if you perform

a search based on Option Switch 31, Position 5, the **Options** panel displays the drawing property that meets the specified criteria.



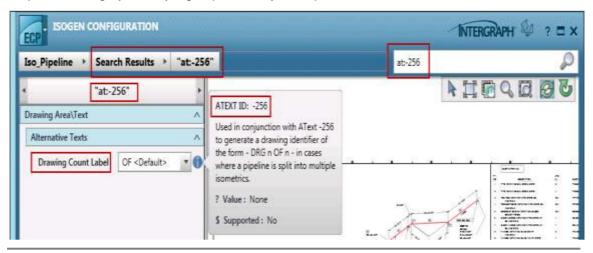
Property Name Search

To perform a property name search, the software requires that you type a text string that contains any part of the original I-Configure property name in the **Search** box. For example, if you search for **weight**, the software locates all of the property names that include the text string **weight**, and then displays the results in the **Options** panel.



AText Number

You can use the **Search** box to locate in which Alternative Texts collection a specific AText number belongs. For example, if you search for **-256**, the software locates it and displays the respective category and style group in the **Options** panel.



What do you want to do?

- Search by keyword (on page 162)
- Search by option switch (on page 164)
- Search by I-Configure property (on page 165)
- Search by AText number (on page 167)
- View the Search history (on page 168)

Search by keyword

- 1. Type the required text string in the **Search** box. For example, if you are looking for the options group that contains the properties for controlling the variable layout material list format, you can type **variable**.
- 2. Click Search P.

The software searches all of the options group headings and displays the results in the **Options** panel.



NOTE Click \sim to expand the style category, group, and options group headings so that you can view the individual properties. The example below shows the properties that are included in the **Variable** options group for the drawing material list:



Search by option switch

1. Click in the Search box.

The **Search** box displays options for adding a search filter.



2. Select Option Switch.

Options display for specifying the option switch and position.

- 3. In the **Option Switch** box, type the option switch number.
- 4. Under **Positions**, select the option switch position, if needed.

The defined search criteria displays in the **Search** box. In the example below, a search definition is created to locate the drawing property that defines Option Switch 31, Position 2.



NOTE Select as many switch positions as needed.

5. Click Search ...

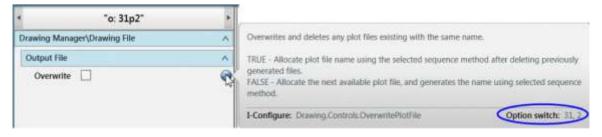
When the search is complete, the software displays the results in the **Options** panel.

6. To view the property, click — to expand the category, group, and option headings. The example below shows the search results for the isometric drawing property that defines Option Switch 31, Position 2:



■ NOTES

To view a property's option switch information, mouseover Tool Information in the Options panel.



You can also perform a manual search to locate a property based on the option switch. In the **Search** box, type o: followed by the option switch number and, optionally, the switch position (for example, o:31p2), and then click **Search**. Use a comma to separate multiple option switch positions, for example, o:31p2,3,4.

Search by I-Configure property

1. Click in the Search box.

The **Search** box displays options for adding a search filter.



2. Select I-Configure

The search filter ic: displays in the Search box.

3. Type a text string that contains a portion of the property name. For example, if you are looking for properties that control the accumulation of components on the material list, you can type **accumulation**.



4. Click Search .

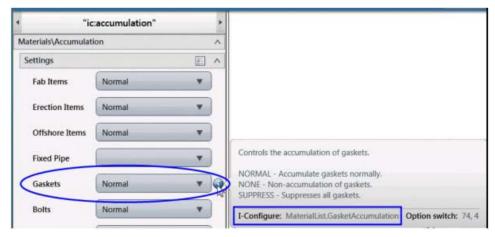
When the search is complete, the software displays the results in the Options panel.

5. To view the each property, click — to expand the style category, group, and option headings. The example below shows all of the isometric drawing properties in which accumulation is part of the I-Configure property name:



■ NOTES

- You can also perform a manual search for a property based on its I-Configure property name. In the Search box, type ic: followed by the required text string (for example, ic:accumulation), and then click Search
- To display a description of each property, as well as its I-Configure property name, mouseover Tool Information in the Options panel.



Search by AText number

1. Click in the Search box.

The **Search** box displays options for adding a search filter.



2. Select AText.

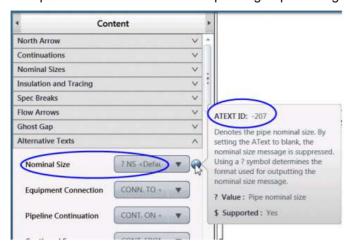
The search filter at: displays in the Search box.

- 3. Type the AText number to locate. For example, type -207.
- 4. Click **Search** \bigcirc .

When the search is complete, the category and style group that contain the AText number display in the **Options** panel.



5. To locate the AText, navigate to the specified category and style group. In the example above, use the **Navigation** bar to navigate to **Drawing Area > Content**, and then click to expand the **Alternative Texts** options group heading.



■ NOTES

- To view a property's AText number, mouseover Tool Information in the Options panel.
- You can also perform a manual search to locate a specific AText number. In the **Search** box, type **at:** followed by the required number (for example, **at:207**), and then click **Search**.

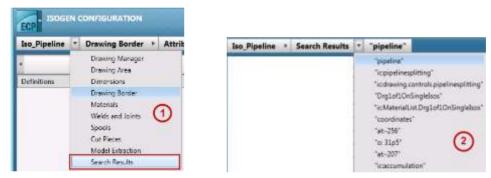
View the Search history

Use one of the methods listed below to view a history of all the searches that have been performed in the current session.

- **★ IMPORTANT** When you exit **Isogen Configuration**, the search history for that session is erased.
- Click once in the **Search** box. The box expands to show a list of previous searches:



Click the Style Name list on the Navigation bar, and then select Search Results (1). Next, open the Search Results list to display the current search history (2).



NOTE To review the results from a previous search, select the search criteria in the **Search Results** list. The **Options** panel updates to display the search results.

Configure Drawing and Report Settings

Drawings and Reports enables you to set properties for isometric drawing output, such as dimension styles, layers, drawing frame text, material lists, and welds.

What do you want to do?

- Customize isometric style options (on page 171)
- Import data from a style XML file (on page 173)
- Export an isometric drawing style fragment (on page 176)
- Create a dotted dimensioned label (on page 182)
- Assign labels (on page 184)
- Populate the title block (on page 185)
- Set drawing frame options (on page 187)
- Edit the North arrow on isometric drawings (on page 188)
- Set drawing dimension options (on page 189)
- Control drawing content (on page 189)
- Set drawing control options (on page 191)
- Specify drawing format (on page 191)
- Create a key point note (on page 191)
- Enable revision clouds (on page 192)
- Assign unique part numbers (on page 194)
- Override isometric drawing style options (on page 196)
- Select a font for isometric drawing extraction (on page 199)
- Define a dynamic font for isometric drawing extraction (on page 199)
- Place barcode data on the isometric drawing (on page 200)
- Set flow arrow options (on page 203)
- Set up a material list (on page 209)
- Set up a cut list (on page 211)
- Specify a label for the material list (on page 210)
- Display a pipeline list (on page 212)
- Print welds (on page 213)

- Specify a label for the weld list (on page 213)
- Map isometric data to drawing layers (on page 215)
- Add a detail sketch (on page 217)
- Configure the software to use an ASCII symbol file (on page 219)
- Create a custom symbol for isometric drawings (on page 220)
- Use piping isometric change management (on page 223)

Customize isometric style options

Isometric drawing styles control several aspects of the isometric drawing output, including the output location and the type of object used for drawing creation. Each drawing style is also associated with a set of options.

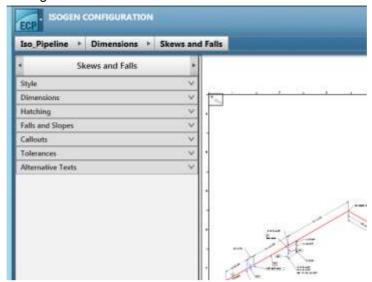
- 1. Right-click an Isogen isometric component or package in the **Management Console** or **Drawing Console**, and then select **Edit Options** to open **Isogen Configuration**.
- 2. On the **Home** screen, select a style group from one of the style category panes. For example, to set properties that define the display and format of dimensions for skews and falls, select **Skews and Falls** from the **Dimensions** category pane.



TIP To find a specific option or property, use the **Search** box. For more information, see *Search Filters* (on page 160).

The software switches to the **Configuration** view and displays each options group that belongs to the selected style group. The example below shows the options groups that

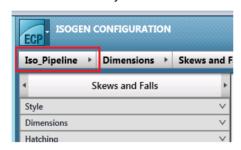
belong to Skews and Falls.



- 3. Expand and collapse each options group heading until you have set all of the needed isometric drawing properties.
 - TIP Use the **Navigation** bar to move through the remaining style categories and groups, and edit other drawing properties as needed. For more information, see *Navigation Techniques* (on page 155).
- 4. Click Save to update the active style file with the new property settings.
 - ★ IMPORTANT You must have write permission to the reference data to save changes to an isometric style.
- 5. Use the **Drawing Preview** panel to review your changes.

■ NOTES

- The drawing preview does not reflect the drawing options that you have set in the Smart 3D category.
- Click the active style breadcrumb in the **Navigation** bar to return to the Home screen.



Alternatively, click im, and then select Home.

Import data from a style XML file

You can import isometric drawing style data from a style XML file directly through **Isogen Configuration**. The **Import** command provides a lot of flexibility for importing style data. For example, you can choose to import the entire contents or only a portion, or *fragment*, of an isometric drawing style file into the current style file. The ability to import fragments of existing styles makes it easy to move around specific drawing properties between isometric drawing style files without the need to create a new .xml file from scratch. Plus, imported style settings are available immediately in the software.

■ NOTE A style fragment specifies a group of properties that you can set or a collection of options that define the columns of a report, such as the user-defined cut list (**Cut Pieces > Drawing Cut List. > User Defined Columns**).

- 1. Right-click an Isogen isometric component or package in the Management Console.
- 2. Select Edit Options to open Isogen Configuration.
- 3. Click the Isogen Configuration menu, and then select Import.
- 4. Browse to the style .xml file, and then click Open.

The software opens the **Import Options from Style** dialog box and displays the properties and style settings contained in the selected .xml file.

- **NOTE** Alternatively, you can browse to and select a style fragment .xml file. In which case, the software bypasses the dialog box and imports the entire style fragment.
- 5. Use the dialog box options to control the display of the file contents in the dialog box and to specify the properties and settings to import. For example, to view report definitions, select **All Reports**. For more information about the dialog box options, see *Import Options from Style Dialog Box* (on page 174).
- 6. Click **OK** to import the selected style data.
 - **Isogen Configuration** reads the content of the style .xml file and applies the specified drawing style properties, Alternative Text settings, and report definitions to the active isometric drawing style. When the import is complete, the software displays a status message at the top of the **Preview** window.
- 7. Click to update the active style .xml file with the selected properties and settings.

 * IMPORTANT You must have write permission to the reference data to save changes to an isometric style.
- 8. Update the drawing to see the new applied styles.
- ▶ NOTE If a style fragment file with the same name and with an .xml extension exists in the same folder as a template file, Isogen automatically loads it. This allows you to set properties specific to the template .xml file, such as margins and list locations.

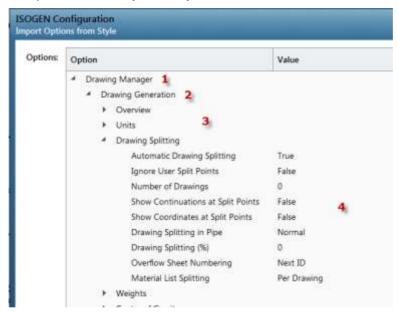
Import Options from Style Dialog Box

Provides options for selecting the drawing properties and settings to be imported from a style .xml file. When you open the dialog box, **Isogen Configuration** displays a hierarchical list of the style categories, style groups, associated options groups, and all respective isometric drawing properties and current settings.

NOTE You use the **All AText**, and **All Reports** check boxes to display Alternative Text (AText) settings and report definitions that are also present in the style .xml file. The **Current** check box is selected by default, which automatically displays all of the properties and settings in the selected style .xml file.

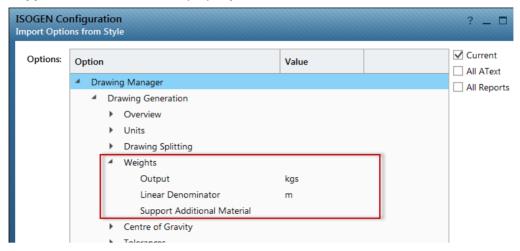
Option

Displays a hierarchical list of the style properties and settings in the selected style .xml file. At the root level is the *style category* (1). Next, is the *style group* (2). Under the style group are groupings of options (3), each of which contains related isometric drawing properties and corresponding property values, if set (4). Use the part and puttons to expand and collapse the hierarchy so that you can view the file contents as needed.

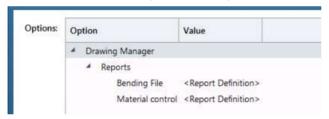


Value

Displays the current setting for a drawing property. If the setting is undefined, no value displays. In the example, **Output** is set to **kgs** and **Linear Denominator** is set to **m**. The **Support Additional Material** property is undefined.



When the property is a table or report, the dialog box displays **<Table>** or **<Report Definition>**, as appropriate. In the example, the report definition for the Bending file and the Material Control file are part of the export list.



★ IMPORTANT The Current, All AText, and All Reports check boxes listed below allow you to control which contents of the selected style .xml file display in the dialog box.

Current

Contains all of the properties and settings available in the selected style .xml file. The software excludes AText settings and reports that cannot be imported individually. This is the default setting.

All AText

Contains all of the AText settings available in the selected style .xml file. You cannot import an individual AText setting. You must import the entire group of AText settings.

All Reports

Contains all of the report definitions available in the selected style .xml file.

Remove

Excludes the selected property, AText setting, or report definition from being imported.

NOTE AText and some reports can be imported only as a group. Removing one item from the group results in the entire group being excluded from import.

Cancel

Closes the dialog box without importing any isometric drawing style data.

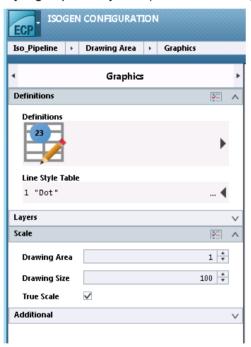
OK

Imports the selected data and settings into the active isometric drawing style. Imported style settings are available immediately in the software. However, to permanently update the active style file with the new settings, you must click Save before exiting **Isogen** Configuration.

Export an isometric drawing style fragment

- 1. Right-click an Isogen isometric component or package in the **Management Console**, and then select **Edit Options** to open **Isogen Configuration**.
- 2. On the Home screen, select the style group that contains the isometric drawing properties to export.

Isogen Configuration switches to the **Configuration** view and displays the **Options** panel. The style group that you selected is now the active stylegroup. In the example, the active style group is **Graphics** (from the **Drawing Area** style category).



3. Click the Isogen Configuration menu and then select Export Style Fragment.

The **Export Style Fragment** dialog box displays and shows the options groups and drawing properties contained within the active style group. If a property is defined, its current setting appears in the **Value** column.

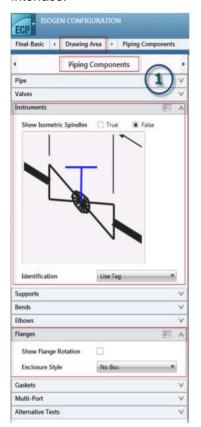
- Use the Current, Changes, All AText, and All Reports check boxes to control the contents
 of the dialog box. For example, to include exportable reports, select All Reports. Likewise,
 to include Alternative Text definitions, select All AText.
 - **NOTE Current** is the default selection, which limits the export data to the properties and current settings for the active style. For more information, see *Export Style Fragment Dialog Box* (on page 177).
- 5. To exclude any object from being exported, select it under **Options**, and then click **Remove**.
- 6. Type the full path location and filename of the .xml file to create during export. Alternatively, click **Browse**, navigate to the folder location, type a name for the .xml file, and then click **Save**.
- 7. Click **OK** to begin exporting the selected isometric drawing style properties and definitions. When the export completes, **Isogen Configuration** displays a status message.

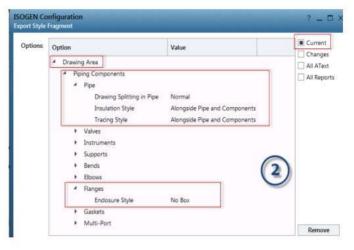
Export Style Fragment Dialog Box

Provides options for selecting the drawing properties and settings to be exported and specifying the output location of the style fragment .xml file. By default, the drawing properties and settings that initially display in the dialog box are based on the *active* style group. The *active* style group is what appears in the **Options** panel when you first open the dialog box. In the example, the active style group is **Piping Components** (1). When you open the **Export Style Fragment** dialog box, **Isogen Configuration** includes a hierarchical list of the active style group, including its associated options groups, each group's respective isometric drawing properties, report definitions (where applicable), and current settings (2).

You use the **Current**, **Changes**, **All AText**, and **All Reports** check boxes to include and exclude the groups of settings to be exported. **Current** is selected by default, which automatically includes all of the properties and settings that belong to the active style group.

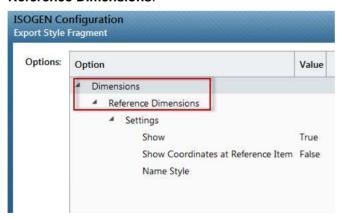
NOTE The root of the displayed hierarchy is always the currently selected style category. In the example, the current style category is **Drawing Area**. For more information, see Review the Interface.





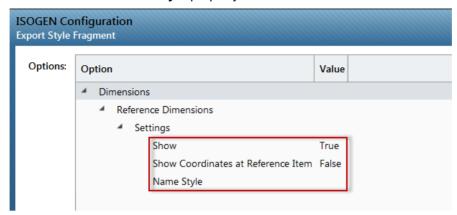
Option

Displays a hierarchical list of the drawing style properties to be exported. The root of this hierarchy is the *style category*. Next, is the *style group*. Under the style group are groupings of options, each of which contains related isometric drawing properties. Use the and abuttons to expand and collapse the hierarchy so that you can view the contents as needed. In the example, the current style category is **Dimensions**, and the active style group is **Reference Dimensions**.



Value

Displays the current setting for a drawing property. If the setting is undefined, no value displays. In the example, **Show** is set to **True** and **Show Coordinates** at **Reference Item** is set to **False**. The **Name Style** property is undefined.



When the property is a table or report, the dialog box displays **<Table>** or **<Report Definition>**, as appropriate. In the example, the report definition for the Bending file and the Material Control file are part of the export list.



★IMPORTANT

- The Current, Changes, All AText, and All Reports check boxes listed below allow you to control which groups of properties and settings are exported to the style fragment .xml file. Each check box exists in one of three states:
 - Unchecked indicates that no properties from that group are included for export.
 - Checked indicates that all properties in the group are included for export.
 - Indeterminate indicates that only some properties in the group are included for export.

Selecting a check box cycles it sequentially through each of the three states in the following order: unchecked, checked, and indeterminate. The table below outlines how moving from one state to another impacts what properties are subsequently included for export:

Moving from to	Resulting export list
Unchecked to Checked	Adds all properties in the selected group to the export list.
Checked to Indeterminate	Removes all properties that belong only to the selected group from the export list.
Indeterminate to Unchecked	Removes all properties that belong to the selected group from the export list. Any property in the selected group that also belongs to another option group, is also removed from the export list. In short, all occurrences of the property are excluded from export.

 Because some properties are contained in multiple options groups, cycling between the check boxes can cause the state of another group to change.

Current

Contains all of the properties and settings in the active style group but excludes all Alternative Text (AText) settings and any report definitions that are either disabled or that cannot be exported individually. When there is no active style group, this option contains all of the properties and (individually exportable) enabled reports in the style category. This is the default setting.

Changes

Contains all of the properties that have been modified since **Isogen Configuration** was opened, excluding all AText settings and any report definitions that cannot be exported individually.

All AText

Contains all of the AText settings. You cannot export an individual AText setting. You must export the entire group of AText settings.

All Reports

Contains all of the report definitions that can be exported individually.

NOTE The reports listed below can be exported individually:

- Bending File report
- Drawing Cross Reference report
- Printed Material List report
- Site Weld Information report
- Weld Summary report
- Neutral report

- Cut List Summary report
- Material Control report
- Repeatability Return report
- Traceability report
- Spool Information report

Remove

Excludes the selected property, AText setting, or report definition from the export.

■ NOTES

- AText and some reports can be exported only as a group. Removing one item from the group also removes any other items that must be exported with it.
- Removing an item from a selected group changes the state to *indeterminate* because all items in the group are no longer included.
- Removing the last item from any group changes the state to unchecked because no items in the group are included.

Path

Specifies the full path location and file name of the .xml file to create during export of the isometric drawing style data.

Cancel

Closes the dialog box without exporting the isometric drawing style data.

OK

Exports the selected options, creates the style fragment .xml file, and saves the file in the specified location.

Set drawing units

1. Click **Home** and then select **Options**.

The Options dialog box displays.

2. Select millimeters or inches in the Drawing Units list.

NOTE Your selection controls only the units of measure that display within the **Options** panel in the **Configuration** view. It does not control the units of measure that are output during generation of the isometric drawing.

3. Click OK.

Create a dotted dimensioned label

Drawing labels provide information about the objects inside a drawing view, such as properties from a database report, text, or graphics (such as a north arrow).

The following workflow demonstrates how to create a label that you can output to an isometric drawing to indicate dotted symbology on specific isometric parts. For more information about dotted symbology drawing options, see *Dotted Symbology* in the *Isometric Drawing Options Reference Guide*. You can access this document using the **Help > Printable Guides** command in the software.

1. In the Catalog task, click Tools > Define Label.

The software opens the **Define Label** dialog box.

2. Select New COM Label

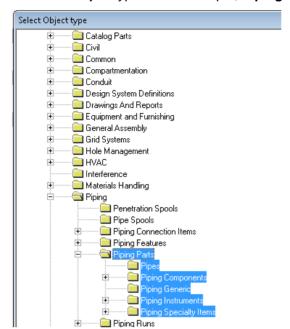
The Label Editor displays.

- 3. Type a name in the **Name** box. For example, type **Dotted dimensioned label**.
- 4. Optionally, type a description in the **Description** box.
- 5. If required, select Name in the Properties grid, and then click Remove X.
- 6. Click Add +

The **Select Properties** dialog box appears so that you can specify the object type and the properties to include.

7. Click More in the Object type used as the basis for the property identification list.

The Select Object Type dialog box opens and displays the Smart 3D hierarchy.

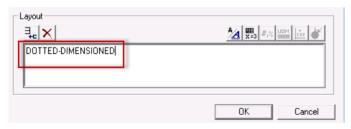


8. Select the object type. In the example, **Piping Parts** is selected.

- 9. Click **OK** to close the dialog box and return to the **Select Properties** dialog box.
- 10. Set **Display properties in this category** to **Standard**, and then select **Reporting Requirement** in the list of property names.
- 11. Click **OK** to close the dialog box and return to the **Label Editor**.
- 12. Select Reporting Requirement in the Properties grid, and then click Insert a field to layout +c.

The Label Editor inserts <F>Reporting Requirement</F> in the Label Text window.

13. Delete <F>Reporting Requirement</F>, and type DOTTED-DIMENSIONED.



NOTE When you generate the drawing, the software the software displays the lines or components in a dotted line.

- 14. Select the typed text in the **Label Text** window, and then click **Block Definition**
 - The **Block** dialog box displays.
- 15. Select **Reporting Requirement** in the **Property** list, select = in the **Operator** list, and set the **Value** to **Not** to be reported.
- Click OK to close the dialog box, and then click OK to create the label and close the Label Editor.

■ NOTES

- A mapped label takes precedence over a mapped filter.
- To create a dotted undimensioned label, follow the steps in this workflow using DOTTED-UNDIMENSIONED in place of DOTTED-DIMENSIONED.
- For more information about using the Label Editor, see Label Editor in the Drawings and Reports Reference Data Guide. You can access this document using the Help > Printable Guides command in the software

Assign labels

You assign labels to attributes within a drawing through **Isogen Configuration**. There are different types of labels that can be applied. The component note, the continuation note, and the nozzle note are used as examples below.

Isometric Component Note

- Right-click an Isogen Isometric Drawing component or package in the Management Console or Drawing Console, and then select Edit Options to open Isogen Configuration.
- 2. Select Label Mapping in the Smart3D category pane.
 - The Label Mapping panel displays.
- 3. Expand the **Notes** options group heading, and then click **Show** to display the **Component Notes** collection grid.
- 4. Set the attributes for the new component note as needed. For example, select **Pipe-Support** from the list in the **Part Type** field.
- 5. In the **Label Name** field, click the ellipsis button to open the **Catalog Label** dialog box. Select a label to use for the component nNote.
- 6. To enclose the component note by a border or bubble, select an option in the **Message Enclosure** list.
- 7. Click Save to update the active style file with the new property settings.
 - ★ IMPORTANT You must have write permission to the reference data to save changes to an isometric style.

Continuation and Nozzle Note

- 1. Right-click an isometric style in the **Console**, and then select **Edit Options** to open **Isogen Configuration**.
- 2. Select Label Mapping in the Smart3D category pane.

The Label Mapping panel displays.

- 3. Expand the Notes options group heading, and then click Show to display the End Connection Notes collection grid. The END-CONNECTION-EQUIPMENT and END-CONNECTION-PIPELINE entries are the labels that give you the nozzle notes and continuation notes.
- 4. Make changes as needed.
- 5. Click Save to update the active style file with the new property settings.
 - ★ IMPORTANT You must have write permission to the reference data to save changes to an isometric style.

NOTE To enable recursive expansion of embedded labels, the report RFM file must set the **ToParse** flag to **Yes**, as in the following example:

```
<DATA
Column="ShortMaterialDescription"
ToParse="yes"
Visible="yes"/>
```

Populate the title block

You can use labels to customize the title block of a drawing. Labels are often used for single pieces of data, such as the approval date or your company name. You can use **Isogen**Configuration to specify options for the appearance and content of the title block.

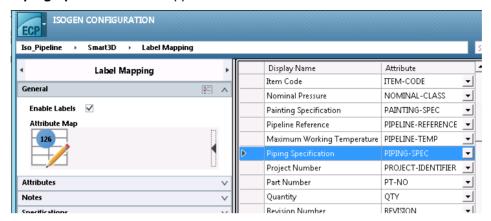
The workflow for setting up attributes to populate the title block involves using **Isogen Configuration** to complete several steps.

- Map host attributes to Isogen attributes using the Smart 3D > Label Mapping > General >
 Attribute Map options.
- Assign labels as needed using the Smart 3D > Label Mapping > Attributes > Drawing Frame Attributes options.
- Position text on the border using the Drawing Border > Attribute Display > Definitions options.

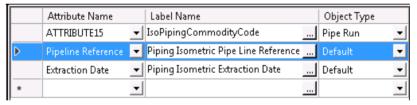
Mapping Attributes

- Right-click an Isogen Isometric Drawing component or package in the Management Console or Drawing Console, and then select Edit Options to open Isogen Configuration.
- 2. Navigate to Smart 3D > Label Mapping > General > Attribute Map.

3. Click **Show** to display the **Attribute Map** grid. Notice the mapping of **Display Name** (host attribute) to **Attribute** (Isogen attribute). For example, the following graphic shows that **Piping Specification** is mapped to PIPING-SPEC.



- TIP You can customize the mapping of Isogen attributes ATTRIBUTE11 through ATTRIBUTE99. ATTRIBUTE1 through ATTRIBUTE10 are reserved by Intergraph. We recommend you start adding your own attributes beginning with ATTRIBUTE21.
- 4. Navigate to Smart 3D > Label Mapping > General > Attributes > Drawing Frame Attributes.
- 5. Click **Show** to display the **Drawing Frame Attributes** grid. Notice the mapping of **Attribute Name** to labels. For example, Isogen attribute PIPELINE-REFERENCE is mapped to the delivered catalog label **Piping Isometrics Pipe Line Reference**.



NOTE Some of the Isogen attributes are hard-coded. For example, **PIPING-SPEC** always returns the specification of the pipeline. It does not need or accept a label.

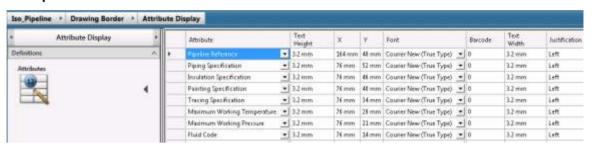
Assigning Labels within the Title Block

- 1. Navigate to Smart 3D > Label Mapping > General > Attributes > Drawing Frame Attributes.
- 2. Specify the mapping of **Attribute Name** to **Label Names**. In the **Label Name** column, click the ellipsis button to display the **Catalog Label** dialog box.
- 3. Expand branches of the catalog label hierarchy and select the label you want to assign to the attribute.

Positioning Labels on the Drawing Border

- 1. Navigate to Drawing Border > Attribute Display > Definitions.
- 2. Click **Show** to display the **Attributes** grid. Notice the mapping of **Attribute** to a location on the drawing sheet. The following graphic shows a sample mapping.

■ NOTE When editing **Drawing Border > Attribute Display > Definitions**, do not remove the **Pipeline Reference** from the **Attribute** list.



- 3. All values are listed in **mm**, measured from the origin of the sheet. For example, **Pipeline Reference** is shown in **3.2 mm font size at X=164mm**.
- 4. Change values as needed to position labels on the drawing border.

Set drawing frame options

Using the Smart 3D > Label Mapping > General > Attribute Map and Drawing Border > Attribute Display > Definitions options, you can specify the content and placement of drawing frame attributes on isometric drawings. Drawing frame text can include revision control information, process conditions, and miscellaneous design or specification notes, placed in the isometric drawing border or title block area. For more information on drawing frame options, see Drawing Border.

To specify this text, you first map Isogen attributes with user-specified text strings. You will use the text strings during the remainder of the attribute definition process. Then, you specify the size and position of the attribute text in the **Drawing Border > Attribute Display > Definitions > Attributes** collection.

For example, you can map the Isogen attributes <code>DRG</code> and <code>DESCRIPTION</code> with the strings <code>Drawing Number</code> and <code>Description</code>, respectively. When Isogen processes the Piping Component File (PCF) content given below, the values "CW-PipeRun1" and "Chillwater Drawing" are inserted into the title block accordingly.

```
DRG CW-PipeRun1
DESCRIPTION Chillwater Drawing
```

You can also use labels to specify drawing frame text. The **Smart 3D > Label Mapping > Attributes > Drawing Frame Attributes** options enable you to map Isogen attributes to label definitions in the catalog.

1. Right-click an Isogen Isometric Drawing component or package in the **Management Console** or **Drawing Console**, and then select **Edit Options**.

Isogen Configuration opens.

2. Navigate to Smart 3D > Label Mapping > General > Attribute Map.

3. Click **Show** to display the **Attribute Map** properties grid.

TIPS

- Type a meaningful text string in the **Display Name** column. You will use this text string
 in the **Drawing Border** style category.
- Select corresponding Isogen attribute in the Attribute column.
- 4. Navigate to **Drawing Border > Attribute Display > Definitions > Attributes**.
- 5. List all of the attributes along with their corresponding locations and text sizes. Use the **Display Name** strings that you typed in the **Attribute Map** grid.
- To use a label for drawing frame text, add an Isogen attribute to the **Drawing Border** >
 Attribute Display > Definitions > Attributes collection, and then map it to a label name and message enclosure.

NOTES

- When editing the Drawing Border > Attribute Display > Definitions > Attributes collection, do not remove Pipeline Reference from the Attribute list.
- Sample labels for drawing frame text on isometric drawings include a pipeline reference label and current date label. The templates for these labels are located in the [Product Folder]\CatalogData\Symbols\Labels\Base Templates folder.
- Another example of drawing frame customization is removing the north arrow. For more information, see Edit the North arrow on isometric drawings.
- You can customize the mapping of Isogen attributes ATTRIBUTE11 through ATTRIBUTE199. ATTRIBUTE1 through ATTRIBUTE10 are reserved by Intergraph. We recommend that you start adding your own attributes beginning with ATTRIBUTE21.

Edit the North arrow on isometric drawings

- 1. Right-click an Isogen Isometric Drawing component or package in the **Management Console** or **Drawing Console**, and then select **Edit Options**.
 - Isogen Configuration opens.
- 2. On the Home screen, select **Attribute Display** in the **Drawing Border** category pane. Alternatively, use the **Navigation** bar to navigate to the required style category and group. For more information, see *Navigation Techniques* (on page 155).
 - The software switches to the **Configuration** view and displays the contents of the **Attribute Display** panel. The **Attribute Display** panel contains a single collection of options called **Definitions**.
- 3. Expand the **Definitions** heading, and then click **Show** to open the **Attributes** collection data grid.
- 4. In the Attribute column, select NORTH-ARROW-POSITION.
- 5. In the **X** and **Y** boxes, specify the X- and Y-coordinates for the north arrow.
 - TIP To turn the north arrow off, type 0 in the XPos and YPos boxes.
- 6. In the **Text Height** and **Text Width** boxes, specify the height and width.

- 7. Click saye to update the active style file with the new property settings.
 - ★ IMPORTANT You must have write permission to the reference data to save changes to an isometric style.
- 8. Use the **Drawing Preview** panel to review your changes.

■ NOTES

• The number that displays on the **Attributes** icon indicates the number of defined attributes. The example below indicates that 15 attributes are currently defined in the grid.



You do not need to map the North Arrow Symbol attribute in the Smart3D > Label
 Mapping > General > Attribute Map collection grid.

Set drawing dimension options

Using the groups of options in the **Dimensions** category, you can specify how dimensions are used and displayed within the drawing. These options include such things as how dimensions are rounded, limits at which to suppress dimension display, and how coordinates are displayed. For example, to turn on the dimensioning for tapped branches on piping, you do the following:

 On the Home screen, select Piping Components and Features in the Dimensions category pane. Alternatively, use the Navigation bar to navigate to the required style category and group. For more information, see Navigation Techniques (on page 155).

The software switches to the **Configuration** view and displays the contents of the **Piping Components and Features** panel.

- 2. Expand Tap Branches.
- 3. Set Tap On Pipe to one of the following settings: None, Full, or Pipe Only.
- 4. Click Save to update the active style file with the new property settings.
 - ★ IMPORTANT You must have write permission to the reference data to save changes to an isometric style.
- 5. Use the **Drawing Preview** panel to review your changes.
- **NOTE** For more information about the drawing dimension properties, see *Dimensions* in *Isometric Drawing Options Reference Guide*, which is available using the **Help > Printable Guides** command in the software.

Control drawing content

Using the **Smart 3D > Content** options, you can specify the content of the isometric drawing. The options allow you to set such things as turning on and off specific coordinates or excluding certain items from the drawing. For descriptions of these options, see **Content** in the **Smart 3D** section of the *Isometric Drawing Options Reference Guide*, available with the **Help > Printable Guides** command in the software.

For example, if you want to exclude continuation parts from the drawing, you would set the following:

1. Right-click an Isogen Isometric Drawing component or package in the **Management Console** or **Drawing Console**, and then select **Edit Options**.

Isogen Configuration opens.

2. Select **Content** in the **Smart 3D** category pane. Alternatively, use the **Navigation** bar to navigate to the required style category and group. For more information, see *Navigation Techniques* (on page 155).

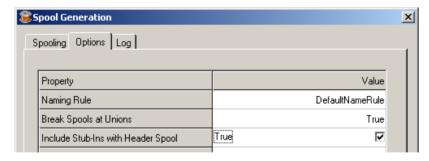
The software switches to the **Configuration** view and displays the contents of the **Drawing** panel.

- 3. Expand the **General** options heading, and then select the **Exclude Continuation Parts** check box.
- 4. Click software excludes the active style file with the new property settings. Notice that the software excludes the continuation parts from the drawing content.
 - ★ IMPORTANT You must have write permission to the reference data to save changes to an isometric style.

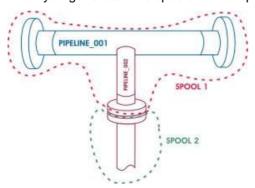
Using the Honor Spool At Branch Option

Another way you can control the content of your isometric drawings is to create your drawings on pipelines that maintain spool integrity at branches. The **Smart 3D > Content > General > Honor Spool At Branch** option gives you this control.

You should generate the spools with the **Include Stub-Ins with Header Spool** property set to **True** when using the **Honor Spool At Branch** option. For more information, see the *Piping User's Guide*, available from **Help > Printable Guides**.



For example, in the following graphic, even though the software extracts Pipeline_001 and the stub-in and flange do not belong to the pipeline, they must be manufactured together because they are part of the same spool. Selecting the **Honor Spool At Branch** option means that all of Spool 1 is included in the isometric drawing for Pipeline_001 and is not included in the isometric drawing for Pipeline_002 (provided the **Include Stub-Ins with Header Spool** was set to **True** when you generated the spools for both pipelines).



For more information on this option, see Content.

Set drawing control options

You use the **Drawing Manager** options to set system controls for the isometric drawing. For more information about drawing control options, see *Drawing Manager* in the *Isometric Drawing Options Reference Guide*. You can access this document using the **Help > Printable Guides** command in the software.

Specify drawing format

The **Drawing Area** style category in **Isogen Configuration** contains options for defining drawing isometric drawing objects, such as components, enclosures, and details sketches. For example, you can do the following:

- Specify the style of the enclosure box for flange rotation angles as having round ends with the Drawing Area > Piping Components > Flanges > Enclosure Style option.
- Set Drawing Area > Content > North Arrow > Show Enclosure to True to display the north arrow inside its own box.

For more information about drawing format options, see *Drawing Area* in the *Isometric Drawing Options Reference Guide* available with the **Help > Printable Guides** command in the software.

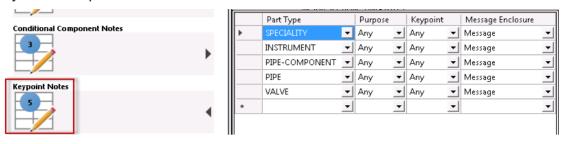
Create a key point note

- 1. Right-click an Isometric Isometric Drawing component or package in the **Management Console** or **Drawing Console**, and then select **Edit Options**.
 - Isogen Configuration opens.
- 2. On the Home screen, select Label Mapping in the Smart 3D pane.

The software switches to the **Configuration** view and displays the contents of the **Label Mapping** panel.

3. Expand the **Notes** options group heading, locate **Keypoint Notes**, and then click **Show** to open the **Keypoint Notes** collection data grid.

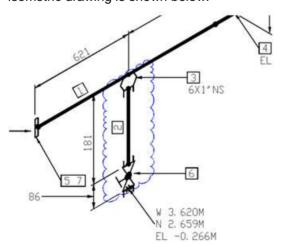
The right side of the window shows the options defined for key point notes within the current style. An example is shown below.



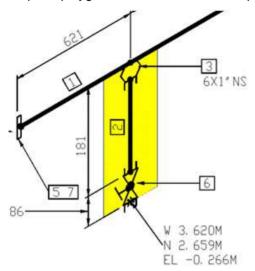
- 4. Use the empty row to create a new key point note, setting the attributes for the note as needed.
- 5. Specify a purpose for the note in the Purpose list.
- 6. In the **Keypoint** list, select the required key point. If a specific key point is not needed, select **Any**.
- 7. To enclose the note by a border, select the required type in the Message Enclosure list.
- 8. Click Save to update the active style file with the new property settings.
 - ★ IMPORTANT You must have write permission to the reference data to save changes to an isometric style.
- ▶ NOTE For information about assigning a key point note to a drawing component, see *Insert a Note* in the *Common User's Guide*, which is available using the **Help > Printable Guides** command in the software.

Enable revision clouds

A revision cloud is an Isogen customization feature that typically indicates a section of the pipeline that has been revised. An example of a traditional revision cloud as it appears on the isometric drawing is shown below:



The traditional representation of revision clouds on the isometric drawing can be modified to output a polygon enclosure, which can optionally be filled with a color.



NOTE There are two revision attributes. The first is the pipeline revision attribute, which defines the current revision of the whole pipeline. This attribute is commonly incremented whenever a new revision is issued. The second is a component revision attribute. This attribute is available for all components in the pipeline and defines the revision that the component was introduced in the pipeline. For revision clouds to be activated on specific components, the pipeline revision and the component revision must be equal.

1. On the Home screen, select **Revision Clouds** in the **Drawing Area** category pane. Alternatively, use the **Navigation** bar to navigate to the required style category and group. For more information, see *Navigation Techniques* (on page 155).

The software switches to the **Configuration** view and displays the contents of the **Revision Clouds** panel.

- 2. Expand Settings.
- 3. Set Enabled to True.
- 4. Use the **Distance** box to specify the distance from the component centerline to the revision box.
- 5. Define the required values for **Colour** and **Layer**. The numbers you specify depend on the drawing system being used, such as AutoCAD or MicroStation.

6. Use **Enclosure Style to** specify whether the revision box enclosure is a cloud or a polygon.

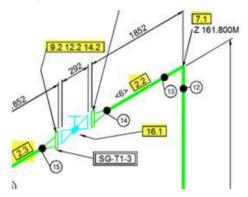
■ NOTES

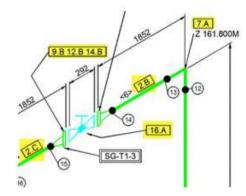
- If you select Cloud, you must also define Minimum Cloud Arc and Maximum Cloud Arc.
- If you select **Polygon**, you can optionally define a fill color for the enclosure.
- 7. Click Save to update the active style file with the new property settings.
 - ★ IMPORTANT You must have write permission to the reference data to save changes to an isometric style.
- 8. Use the **Drawing Preview** panel to review your changes.

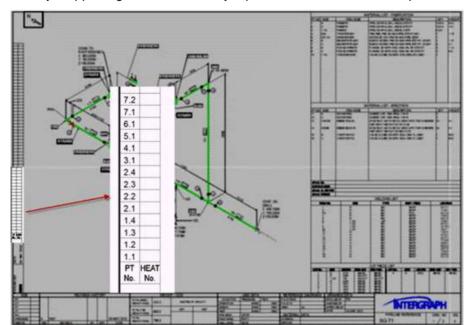
Assign unique part numbers

Unique numbers are applied to each part on the isometric drawing. This is typically used for traceability and to assign heat numbers.

The part numbers are unique per pipeline and are in the format of **1.x**, where *x* is a number or an alpha character. The examples below show the two format options (highlighted in yellow).







A table of unique part numbers can also be plotted on the isometric drawing. This table is usually mapped against the manually-input heat numbers. An example is shown below:

On the Home screen, select **Part Numbers** in the **Materials** category pane. Alternatively, use the **Navigation** bar to navigate to the required style category and group. For more information, see *Navigation Techniques* (on page 155).

The software switches to the **Configuration** view and displays the contents of the **Part Numbers** panel.

- 2. Expand Format.
- 3. Set Unique Part Number to Alpha or Numeric, depending on the required suffix.
- 4. Use **Unique Gasket Numbers** and **Unique Bolt Numbers** to control whether gaskets and bolts are assigned unique numbers.
- 5. Set the remaining properties as needed.
- 6. Click Save to update the style file with the new property settings.

NOTE To output and define the contents of a traceability file, see *Define the report layout* (on page 206).

Option Override (Isogen Isometric Drawing Shortcut Menu)

Overrides style settings that have been defined for the selected Isogen isometric drawing. This feature allows you to modify, or even nullify, individual style settings on a per drawing basis without having to create a new style for a single drawing. When you use this command, the software creates a style fragment that is added to the database. When you update the drawing, the style fragment is merged with the isometric drawing style to produce the final style that is used to update the drawing. After the drawing is updated, you can view the style fragment in the *Extraction Data dialog box* (on page 116).

You can access this command by right-clicking an Isogen isometric drawing in the Detail View.

Option Override Dialog Box (on page 197)

Select Options Dialog Box (on page 197)

Override isometric drawing style options

1. Select an Isogen Isometric Drawing component or package in the **Management Console** or **Drawing Console**.

The software displays a list of isometric drawings in the **Detail View**.

2. In the **Detail View**, right-click an Isogen isometric drawing, and select **Option Override**.

NOTE The **Option Override** command does not display on the shortcut menu if multiple drawings are selected.

The **Option Override** dialog box displays.

3. Click Add.

The **Select Options** dialog box displays.

4. Select the isometric drawing options that you want to override, and click **Add**. Hold down the CRTL or SHIFT keys to select multiple options.

■ NOTES

- To search for a particular option by name, type any part of the name in the Search box.
 For example, if you type north, the Properties list updates to display only the Drawing Manager > Drawing Generation > Overview > North Direction and Drawing Area > Content > North Arrow > Show Enclosure options.
- If the **Search** box is empty, all available options are shown in the dialog box.

The software adds the selected isometric drawing options to the list of overridden options in the **Option Override** dialog box.

Click Done.

The **Select Options** dialog box closes.

6. In the **Option Override** dialog box, change the values of the listed options as needed. When you are finished, click **OK**.

The software creates the style fragment, the **Option Override** dialog box closes, and you return to the **Management Console** view.

7. Right-click the Isogen isometric drawing in the Detail View, and then select Update Now.

The software applies the style changes to the drawing.

NOTE After the drawing is updated, you can view the resulting style fragment in the **View Extraction Data** dialog box. For more information, see View style fragment data (on page 115).

Option Override Dialog Box

Displays all of the options that are currently overridden for the selected drawing. You can change the value of any of the options that are listed. You access this dialog box by right-clicking an Isogen isometric drawing in the **Detail View** and selecting **Option Override**.

NOTE If the drawing has no overridden options, the dialog box is empty.

Property

Displays the name of the option. This field is read-only.

Value

Specifies the value of the property. The current value displays by default. Change the value as needed.

Add

Opens the **Select Options** *dialog box* (on page 197), in which you can specify which options are overridden.

Remove

Removes the selected property from the list of overridden options.

OK

Creates an override style fragment.

Cancel

Closes the dialog without making any changes to the Catalog database.

Select Options Dialog Box

Displays all the options that are available and their current settings.

Search

Finds an option based on text that you type. This command searches in the **Property** list. The software returns the results in the table portion of the browser. To find an option, type text in the **Search** box and press ENTER. For example, if you type **s3d**, the **Property** list displays only the options directly associated with Smart 3D and excludes those that are associated with Personal Isogen. You can also type specific option numbers in the **Search** box. For example, you can type **?OPT:79** to find option 79. If the **Search** box is empty, all available options are shown in the dialog box.

Property

Displays the name of the option. You can select multiple options by pressing CTRL or SHIFT while selecting. When you select an individual property in this list, the grid at the bottom of the dialog box displays a brief description.

Value

Displays the current setting of the option. This information is read only.

Add

Adds the selected property to the list of overridden options on the **Override Options** dialog box.

Done

Closes the dialog box, and returns to the **Override Options** dialog box.

Fonts in Isometric Drawings

You can change the font that displays on the isometric drawing. A font information file (*.fif), which contains a default collection of fonts, is delivered in the workstation setup in the [Product Folder]\DrawingsIsometric\Alias\Core Components folder. You specify this file in the isometric options, and then you can select fonts for the material list and the drawing.

If there is a requirement to use a font that is not available in the standard font information file delivered in the workstation setup, the software supports user-specified dynamic fonts. Isogen recognizes the font definition automatically during isometric drawing generation.

What do you want to do?

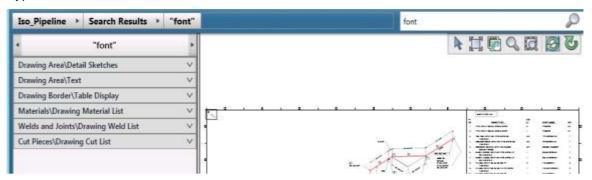
- Select a font for isometric drawing extraction (on page 199)
- Define a dynamic font for isometric drawing extraction (on page 199)
- Place barcode data on the isometric drawing (on page 200)

Select a font for isometric drawing extraction

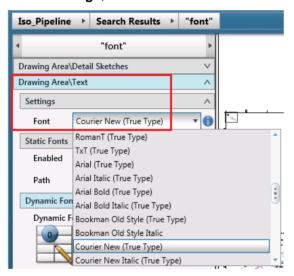
1. Right-click an Isogen Isometric Drawing component or package in the **Management Console** or **Drawing Console**, and then select **Edit Options**.

Isogen Configuration opens.

2. Type **font** in the Search box, and execute the search.



- 3. Expand Drawing Area\Text.
- 4. Under Settings, select a font from the Font list.



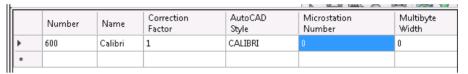
- 5. Click 5ays to update the active style file with the new property settings.
 - ★ IMPORTANT You must have write permission to the reference data to save changes to an isometric style.
- 6. Use the **Drawing Preview** panel to review your changes.

Define a dynamic font for isometric drawing extraction

1. Right-click an Isogen Isometric Drawing component or package in the **Management Console** or **Drawing Console**, and then select **Edit Options**.

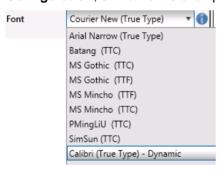
Isogen Configuration opens.

- 2. Navigate to **Drawing Area > Text**, and expand the **Dynamic Fonts** heading.
- 3. Click **Show** to open the **Dynamic Fonts** collection grid.
- 4. Define the needed options. In the example, the font Calibri is defined as a dynamic font.



■ NOTE You must define the **Number** and **Name** properties. For more information about these properties, see *Dynamic Fonts* in the Smart 3D *Isometric Drawing Options Reference Guide*. You can access this document using the **Help > Printable Guides** command in the software.

- 5. Click Saye to update the active style file with the new property settings.
 - ★ IMPORTANT You must have write permission to the reference data to save changes to an isometric style.
- 6. Use the **Drawing Preview** panel to review your changes.
- NOTE After you save the font settings, it displays under the Drawing Area > Text > Font and Materials > Drawing Material List > Settings > Text Font properties in Isogen Configuration, similar to the example below:



Place barcode data on the isometric drawing

The following steps show you how to define a barcode font and then how to configure the isometric drawing frame so that a specific pipeline attribute is output as a barcode.

NOTE If the barcode font you are using is already defined as a dynamic font, skip steps 2-4.

1. Right-click an Isogen Isometric Drawing component or package in the **Management Console** or **Drawing Console**, and then select **Edit Options**.

Isogen Configuration opens.

2. On the Home screen, select **Text** in the **Drawing Area** category pane.

The software switches to the **Configuration** view and displays the contents of the **Text** panel. The **Attribute Display** panel contains a single collection of options called **Definitions**.

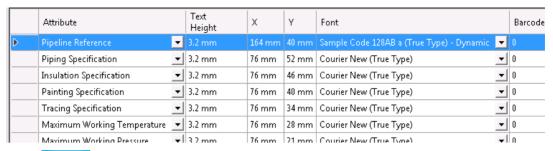
- 3. Expand the **Dynamic Fonts** options group heading, and and then click **Show** to open the **Dynamic Fonts** collection data grid.
- 4. Use the empty row at the bottom of the grid to define the required properties for the barcode font. In the example below, a barcode font called **Sample Code 128AB a** is defined.

	Number	Name	Correction Factor	AutoCAD Style	Microstation Number	Multibyte Width
▶	600	Sample Code 128AB a		Sample Code 128AB a		0
*						

NOTE You must define the **Number** and **Name** properties. For more information about these properties, see **Definitions** under *Attribute Display* in the *Isometric Drawing Options Reference Guide*, which is available with the **Help > Printable Guides** command in the software.

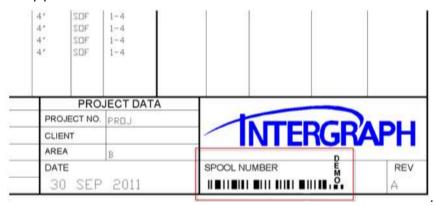
- 5. Click Save to update the active style file with the new property settings.
 - \star IMPORTANT You must have write permission to the reference data to save changes to an isometric style.
- 6. Use the **Drawing Preview** panel to review your changes.
- 7. Navigate to **Drawing Border > Attribute Display**, and then expand the **Definitions** options group heading.
- 8. Click **Show** to open the **Attributes** collection data grid.

 Locate the drawing frame attribute to output as a barcode, and then use the Font list to select the barcode font. In the example below, the pipeline reference attribute is being configured to output using the Sample Code 128AB a font that you defined previously.



- 10. Click Save to update the active style file with the new property settings.
 - ★ IMPORTANT You must have write permission to the reference data to save changes to an isometric style.
- 11. Use the **Drawing Preview** panel to review your changes.
- 12. Close Isogen Configuration.
- 13. In the Drawings and Reports task, use the **Update Now** or **Batch > Update** commands to update the drawing.

The pipeline reference is now shown as a barcode on the isometric drawing.



Flow Arrows

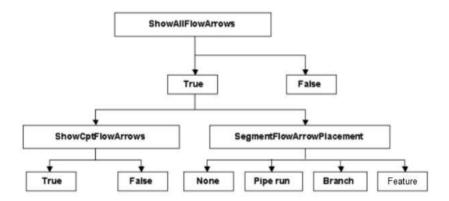
Isometric drawings commonly have annotations to denote the direction of fluid flow through pipe as well as through pipe components. To specify flow direction in the model, switch to the Piping task. Select a pipe run and change the flow direction by clicking the circular icons on the run. Then, use **Isogen Configuration** to define when and where flow arrows are placed on isometric drawings.

Two overall types of flow arrows exist: segment and component. Segment flow arrows display on pipe runs, while component flow arrows display beside components such as valves.

The options for flow arrow placement are located in **Drawing Area > Content > Flow Arrows**. The **Show on Pipe** property toggles all flow arrows on and off. If **Show Flow Arrows** is set to **True**, you can also specify component and segment arrow placement. You can set **Show on Components** so that component flow arrows appear beside components such as valves. Also, you can set **Smart 3D > Content > General > Segment Flow Arrow Placement** to **None**, **Pipe run**, **Branch**, or **Pipe Feature** to denote how arrows are placed on the pipe. The **Pipe run** selection means a flow arrow is placed on each pipe run. The **Branch** selection means a flow arrow is placed on the first leg of each branch, including the header. If the pipeline does not have a branch, then no additional (there should always be at least one) flow arrow is placed even if one pipe run ends and another begins. The **Feature** selection means a flow arrow is placed on each and every feature of a pipe run.

When you configure your isometric styles, you can set **Show on Components** and **Segment Flow Arrow Placement** to match your company specifications. Thereafter, you can control all flow arrows on the isometric drawing by toggling **Show on Pipe** on or off.

The following diagram shows the relationships among the flow arrow options.



Set flow arrow options

Isometric drawings commonly have annotations to denote the direction of fluid flow through pipe and pipe components. You can use **Isogen Configuration** to define when and where flow arrows are placed on isometric drawings.

The properties for flow arrow placement are located in **Drawing Area > Content > Flow Arrows**. The **Show on Pipe** property toggles all flow arrows on and off. If **Show Flow Arrows** is set to **True**, you can also specify arrow placement. You can set **Show on Components** so that component flow arrows appear beside components such as valves. When you configure your isometric styles, you can set **Show on Components** to match your project specifications. Thereafter, you can control all flow arrows on the isometric drawing by toggling **Show on Pipe** on or off.

■ NOTE You cannot set flow arrow options for cable tray or HVAC isometric drawings. This functionality is only applicable to piping isometric drawings.

 On the Home screen, select Content in the Drawing Area category pane. Alternatively, use the Navigation bar to navigate to the required style category and group. For more information, see Navigation Techniques (on page 155).

The software switches to the **Configuration** view and displays the contents of the **Content** panel.

- 2. Expand Flow Arrows.
- 3. Set Show on Components as needed.
 - TIP Set the property to **True** if you want component flow arrows to appear for this style. Set the property to **False** if you do not want component flow arrows to appear for this style.
- 4. Set **Show on Pipe** to **True** or **False**. This property is a master switch that toggles all flow arrows on or off.
- 5. Click Save to update the active style file with the new property settings.
 - ★ IMPORTANT You must have write permission to the reference data to save changes to an isometric style.
- 6. Use the **Drawing Preview** panel to review your changes.

Bending Files

The main purpose of the bending file is to provide an input to a bending machine, which bends the pipe.

Here is an example of a bent pipe and its corresponding bending file information. The pipe configuration is a 2 ft pipe/5 deg bend/2 ft pipe.



	X	Y	Z	Radius	Angle
START	0.0	0.0	0.0	0.0	0.0
BEND1	-607.3	0.0	53.1	0.0	0.0
FINISH	-1216.9	0.0	53.1	0.0	0.0

The START row indicates the start point at 0, 0, 0. The BEND1 row indicates the relative coordinates of the bend location from the start point in millimeters. The distance of 607.3 mm is almost 2 ft (23.909 in). For the FINISH point, the distance of 1216.9 mm is almost 4 ft (47.909 in). The 53.1 mm distance is the absolute elevation between the start and finish points.

You can specify that a bending file is created by selecting **Bending File** in **Drawing Manager > Reports > Report Types** for an isometric style. The keyword <code>BENDING_FILE_APPEND</code> places the bending file information for each extraction consecutively in one file, and the keyword <code>BENDING_FILE_OVERWRITE</code> erases the file and writes to it with each extraction.

Material Lists and Material List Files

The software allows you to specify several different types of material files for isometric drawings. In addition, you can include cut list information. The cut list includes the pieces of cut pipe and their required lengths.

The software contains three different styles for material lists on isometric drawings. Fixed Layout is the default material list for which customization is limited. Variable Layout allows you to specify the attributes in the columns of the material list. User Defined is the most customizable, allowing full control of the attributes in the columns, the number of sections in the material list, and remarks.

★ IMPORTANT The User Defined type of material list is applicable to all isometric drawing types. However, the Variable Layout and Fixed Layout formats are applicable only to piping isometric drawings.

You can also specify labels that correspond to items in the material list.

You can specify the width of a rectangular cross section in a cable tray or an HVAC isometric drawing.

Printed Material List

The printed material list is a text file that contains the same information as the material list on the face of the drawing. The information is continuously appended to the file. You can specify that a printed material list file is created by using **Materials > Drawing Material List**. For more information, see *Set up a material list* (on page 209).

Material Control File

The purpose of material control file is to provide information to a material control system. You can specify this file by using **Drawing Manager > Reports**, and then selecting **Material control** in the **Report Types** list to define the options. For more information about defining reports, see *Define the report layout* (on page 206).

MTO Neutral File

The MTO neutral file is in a plain text format. The purpose of this file is to provide the extracted information to a material control system. You can control the objects included in the neutral file as well as the columns of information.

Optionally, weld and bolt data can appear in the MTO neutral file. The data from these components can appear in columns already existing in the MTO neutral file, or you can specify new columns. You can specify this file by using **Drawing Manager > Reports**, and then selecting **Neutral report** in the **Report Types** list to define the options.

Cut Pipe Report

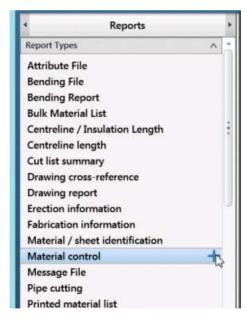
The cut pipe report provides a list of the pieces of cut pipe and their required lengths. To specify options for a cut list, you use the **Cut Pieces > Drawing Cut List** options. For more information, see *Set up a cut list* (on page 211).

What do you want to do?

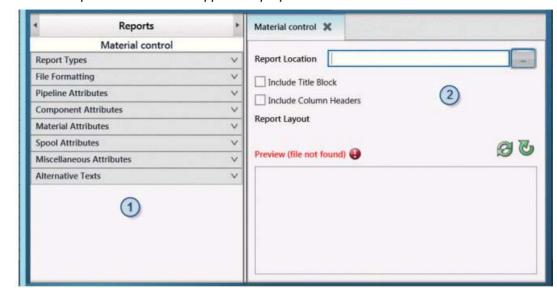
- Define the report layout (on page 206)
- Set up a material list (on page 209)
- Specify a label for the material list (on page 210)
- Set up a cut list (on page 211)

Define the report layout

- 1. Navigate to **Drawing Manager > Reports**.
- 2. Expand **Report Types** in the **Reports** panel, and double-click the report to define. For example, double-click **Material control**.
 - TIP You can also click the green plus sign to add the report to the **Report Definition** view.



The Reports panel updates to show the applicable options groups. The Report Definition



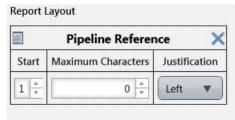
view also updates to show the applicable properties.

- 3. Type the output location and file name for the report in the **Report Location** box. Alternatively, click **Browse**, and navigate to the location.
 - **NOTE** The filename itself can be dynamic. For example, you can use the pipeline reference as part of the filename. For more information, *Dynamic File Naming* in *Report Definition View* (on page 150).

The **Preview (file not found)!** error message disappears. **Isogen Configuration** displays this message until an output location is specified.

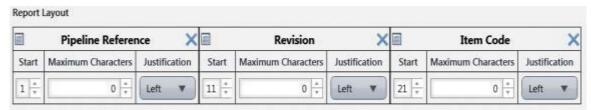
- 4. Expand File Formatting in the Reports panel, and define the properties as needed.
- 5. Use the **Attributes** options groups to specify the columns of attribute data to output in the report. For example, to output the pipeline reference, expand **Pipeline Attributes**, and then double-click **Pipeline Reference**.
 - TIP You can also click **Add Attribute** +, which appears each time you mouse-over an attribute.

The column definition for the attribute data is placed under **Report Layout** in the **Report Definition** view.

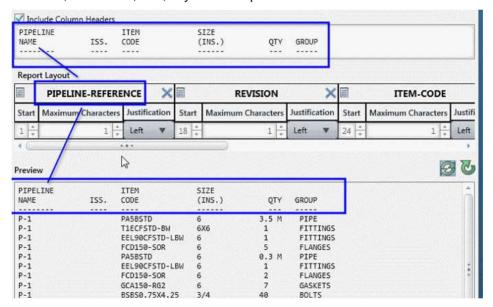


- 6. Define the **Start**, **Maximum Characters**, and **Justification** settings for the attribute data column.
 - **NOTE** Click **Comments** to insert a comment, and type the text in the text box.

7. Continue adding attributes to the report. In the example below, three columns of attribute data are defined for the report.



- TIP To remove a column of attribute data, click Remove Column X.
- 8. To output a title block, click **Include Title Block**, and then type the title.
- 9. To output column headers, click Include Column Headers.
- 10. In the **Column Headers** box, type a display name for each column of attribute data in the report. In the example below, display names are defined for the Pipeline Reference, Revision, Item-Code, N.S, Qty and Group attribute columns.



- TIP Use the SPACEBAR and ENTER keys to add spaces between the display names or for multiple lines of data.
- 11. When finished, click Save

■ NOTES

As you define the report layout, the Preview pane dynamically updates. By default, Automatic Refresh is turned on, so the update occurs automatically. However, if Automatic Refresh is turned off, you must click Refresh to manually update the view in the Preview pane.

- Drawing Manager > Reports organizes all of the reports that Isogen can output during
 isometric drawing generation. In addition to this global Reports panel, the Materials, Welds,
 Spools, and Cut Pieces categories each contain a Reports panel that provides access to a
 content-specific subset of report types.
 - Materials > Reports Material control, Printed material List, and Neutral report
 - Welds > Reports Weld summary and Site weld information
 - Spools > Reports Spool information
 - Cut Pieces > Reports Cut list summary
- You can also use the **Drawing Setup Tool** to define the layout and appearance of the report data in the material list, cut list, or weld list that is plotted on the isometric drawing. Although the **Drawing Setup Tool** contains a subset of the features available in the **Isogen Configuration** environment, its graphical interface makes the initial setup of these three reports easier. For more information, see *Drawing Setup Tool* (on page 234).

Set up a material list

The software contains three different styles for material lists on isometric drawings:

- Fixed Layout is the default material list and is not customizable.
- Variable Layout is a customizable version of the fixed layout style and allows you to specify
 the attributes in the columns of the material list.
- User Defined is the most customizable of the three styles, allowing full control of the attributes in the columns, the number of sections in the material list, and remarks.
- **NOTE** You can also use the **Drawing Setup Tool** to define the layout and appearance of the report data in the material list that is plotted on the isometric drawing. Although the **Drawing Setup Tool** contains a subset of the features available in the **Isogen Configuration** environment, its graphical interface makes the initial material list setup easier. For more information, see *Drawing Setup Tool* (on page 234).
- 1. Right-click an Isogen Isometric Drawing component or package in the **Management** Console or Drawing Console, and then select Edit Options.
 - **Isogen Configuration** opens.
- 2. On the Home screen, select **Drawing Material List** in the **Materials** category pane. Alternatively, use the **Navigation** bar to navigate to the required style category and group. For more information, see *Navigation Techniques* (on page 155).
 - The software switches to the **Configuration** view and displays the contents of the **Drawing Material List** panel.
- 3. Expand the options group that corresponds to the required material list layout: **Fixed**, **Variable**, or **User Defined**.
 - TIP If Visible under Symbol Table in Drawing Border > Table Display is not selected, you must use the fixed layout material list style. Only a fixed layout material list can be used with an Isogen drawing frame.

- 4. Set the properties for the selected material list layout
- 5. Click Save to update the active style file with the new property settings.
 - ★IMPORTANT You must have write permission to the reference data to save changes to an isometric style.
- 6. Use the **Drawing Preview** panel to review your changes.
- 7. Define the report layout (on page 206).

■ NOTES

- The amount of area on the drawing that is reserved for the material list must be set properly, or overlap can occur between the piping symbology and the text of the material list. You can set the reserved area for the material list by setting the Reserved Area Material List in Drawing Manager > Drawing File > Drawing Area Size.
- By default, bolts and gaskets are accumulated automatically in the material list by diameter.
 You can deactivate this feature by setting Bolts or Gaskets to Suppress in Materials >
 Accumulation > Settings.
- A material list can contain cut list information. The cut list includes the pieces of cut pipe and their required lengths. You can specify summary files for the material list and cut list.
- The printed material list is a text file that contains the same information as the material list on the face of the drawing. The information is continuously appended to the file. You can specify that a printed material list file be created using Materials > Reports > Report Types and selecting Printed material list. For more information, see Define the report layout (on page 206).

Specify a label for the material list

- 1. Right-click a Piping Isometric Drawing component or package in the **Management Console** or **Drawing Console**, and then select **Edit Options** to open **Isogen Configuration**.
- 2. On the Home screen, select Label Mapping in the Smart 3D category pane.
 - The software switches to the **Configuration** view and displays the contents of the **Labels** panel.
- 3. Expand the **Attributes** options group heading, and then click **Show** to open the **Component Attributes** collection data grid.
- 4. Specify a setting for **Type** and **Attribute Name**, and then click the browse button under **Label Name** to select an isometric label from the catalog.
- Navigate to Materials > Drawing Material List > Settings, and then set Active List to UserDefined.
- 6. Expand the **User Defined Columns** options group heading, and then click **Show** to open the **Columns** collection data grid.

- 7. In the blank row, define the location for the attribute using the **offset** and **Maximum**Characters columns.
- 8. Click Save to update the active style file with the new property settings.
 - **★ IMPORTANT** You must have write permission to the reference data to save changes to an isometric style.

Set up a cut list

The cut pipe report provides a list of the pieces of cut pipe and their required lengths.

- 1. Right-click an Isogen Isometric Drawing component or package in the **Management Console** or **Drawing Console**, and then select **Edit Options**.
 - Isogen Configuration opens.
- 2. On the Home screen, select **Drawing Cut List** in the **Cut Pieces** pane. Alternatively, use the **Navigation** bar to navigate to the required style category and group. For more information, see *Navigation Techniques* (on page 155).

The software switches to the Configuration view and displays the Drawing Cut List panel.

Expand the option group heading that corresponds to the required cut list layout: Fixed or User Defined.

TIPS

- Fixed is the default cut list and is not customizable.
- User Defined is a customizable version of the cut list.
- 4. Set the properties for the selected cut list layout.
- 5. Click Save to update the style file with the new property settings.

NOTE You can also use the **Drawing Setup Tool** to define the layout and appearance of the report data in the cut list that is plotted on the isometric drawing. Although the **Drawing Setup Tool** contains a subset of the features available in the **Isogen Configuration** environment, its graphical interface makes the initial cut list setup easier. For more information, see *Drawing Setup Tool* (on page 234).

Pipeline Lists

You can specify an option to display process or design information from each pipe run on the isometric drawing. This pipeline list is an embedded Excel workbook.

PNOTE When defining embedded report layout (sizing of columns and rows), consider the report usage first. Because of a Microsoft limitation concerning the size of Windows metafile objects within other applications, the data displayed may be incomplete. Therefore, no column should be out of screen when using 100 percent zoom for the report. Otherwise some columns are ignored when the report is embedded within the drawing. The same limitation exists for rows. To preserve the maximum number of rows displayed, the total header row(s) height should be a minimum of the overall report. Using Microsoft Excel default settings, the maximum number of columns is approximately 20 and the maximum number of rows is approximately 75 (including header rows). For more information on setting the defaults in Microsoft Excel, see your Microsoft Excel documentation.

Display a pipeline list

1. Right-click an Isogen Isometric Drawing component or package in the **Console**, and the select **Edit Options**.

Isogen Configuration opens.

- 2. On the **Home** screen, select **Post Isogen** on the **Smart3D** category pane.
 - The software switches to the **Configuration** view and displays the **Post Isogen** panel.
- 3. Expand the Pipeline List options group heading, and select Show Pipeline List Box.
- Click Save to update the active style file with the new property settings.
 - ★ IMPORTANT You must have write permission to the reference data to save changes to an isometric style.

See Also

Edit Options (Isogen Isometric Drawing Component Shortcut Menu) (on page 134)

Weld Lists and Weld Files

To enable weld list output on an isometric drawing, you must configure a few items in the software.

First, you must map each type of weld to a symbol key (SKEY). You can complete this mapping by opening **Isogen Configuration** for an isometric style and using the **Smart3D > Drawing Creation > Symbol Map > Symbol Map** group of options.

Second, you must set options to show the weld symbols as well as the weld numbers on the isometric drawing. The **Welds and Joints > Generation and Display > Controls > Show Welds** option indicates whether the weld symbol displays on the drawing, and the **Welds and Joints > Enclosures > Show Weld Numbers** option indicates if the weld number displays.

You can also specify that a weld list display on the drawing. The columns shown in the weld list can be customized. The weld list does not display if the option setting to display weld numbers is not also enabled. If needed, you can specify that the weld list information is also saved in a summary file.

The weld file contains the same information as the weld list on the isometric drawing. You can specify a weld file using **Welds and Joints > Reports > Report Types** to enable and define the Weld summary report.

What do you want to do?

- Print welds (on page 213)
- Specify a label for the weld list (on page 213)

Print welds

- 1. Right-click an Isogen Isometric Drawing component or package in the **Management** Console or Drawing Console, and then select Edit Options to open Isogen Configuration.
- On the Home screen, select Generation and Display in the Welds and Joints category pane.
 - The software switches to the **Configuration** view and displays the contents of the **Generation and Display** panel.
- 3. Expand the **Controls** options group heading, and set **Show Welds** to **True**.
- 4. Navigate to **Welds and Joints > Enclosures**, and expand the **Controls** options group heading.
- 5. Set Show Weld Numbers to True.
- To display a weld list on the drawing, navigate to **Drawing Weld > Settings**, and select **Weld List**.

NOTE You can save the weld list information to a file by specifying the Weld summary file options. For more information, see *Reports* in the *Isometric Drawing Options Reference Guide*.

Specify a label for the weld list

- Right-click an Isogen Isometric Drawing component or package in the Management Console or Drawing Console, and then select Edit Options to open Isogen Configuration.
- 2. On the **Home** screen, select **Label Mapping** in the **Smart3D** category pane.
 - The software switches to the **Configuration** view and displays the contents of the **Label Mapping** panel.
- 3. Expand the **Attributes** options group heading, then click **Show** to open the **Weld Attributes** collection data grid.
- 4. Select an attribute in the Attribute Name list.

- 5. Under **Label Name**, click the browse button, and then select an isometric label from the catalog.
- Navigate to Welds and Joints > Drawing Weld List > Settings, and then set Active List to UserDefined.
- 7. Expand the **User Defined Columns** options group heading, and then click **Show** to open the **Columns** collection data grid.
- 8. Define the location for the attribute you added using the offset and MaxChars columns.
- 9. Click Saye to update the active style file with the new property settings.
 - ★ IMPORTANT You must have write permission to the reference data to save changes to an isometric style.
- 10. Use the **Drawing Preview** panel to review your changes.

See Also

Edit Options (Isogen Isometric Drawing Component Shortcut Menu) (on page 134)

Layers

Layers are used to distinguish between graphics within a template or drawing. You can create layers in **SmartSketch Drawing Editor** with the **Tools > Layers** command.

Objects like drawing views, key plan views, report views, and drawing property labels are placed on the **DwgTemplate** layer when you save the drawing document. You should not place manual markups on the **DwgTemplate** layer.

When placing manual markups, such as graphics or company logos, place them in paper space within the drawing layers, not in *model space* (inside a drawing view). Markups placed within *model space* are not preserved. When a graphic rule does not specify a layer, intelligent graphics are placed on the **Default** layer.

Composed Drawings

When you create composed drawings, you can place manual markups on any layer of the drawing.

Volume (Spatial) and Orthographic Drawings by Query Drawings

For volume drawings, manual markups are preserved on any layer of the drawing, except the **DwgTemplate** layer. The **DwgTemplate** layer is reserved for system use. If you placed manual markups on the **DwgTemplate** layer, they will be lost when you update the drawing document.

Isogen Isometric Drawings

If you intend to create manual markups within an Isogen Isometric drawing, place your markups on the **Default** layer. The software preserves this layers when you update drawings. Other layers are not preserved.

If named layers do not exist in the template, the software creates them using the symbology specified in the style XML file. In ISOGEN Configuration, use the options in the **Drawing Area > Graphics > Layers** style group to create new layers within the style XML file. Map definitions to the layers under **Drawing Area > Graphics > Definitions**.

If the named layers do exist in the template, use **Tools > Display Manager** in **SmartSketch Drawing Editor** to change the symbology used within the template.

Map isometric data to drawing layers

You use ISOGEN Configuration to map layers within a previously created drawing border file to isometric data. The options used to define the mapping are found in the **Drawing Area** > **Graphics** > **Layers** and **Drawing Area** > **Graphics** > **Definitions** style groups.

CAUTION Layer definitions created with **Tools > Display Manager** in SmartSketch Drawing Editor override these settings.

- 1. Right-click an Isogen Isometric Drawing component or package in the **Management** Console or Drawing Console, and then select Edit Options.
 - Isogen Configuration opens.
- 2. On the Home screen, select **Graphics** in the **Drawing Area** category pane.
 - The software switches to the **Configuration** view and displays the contents of the **Graphics** panel.
- 3. Expand the **Layers** group, and then click **Show** to open the **Layers** grid.
- 4. Define the settings as needed. The **Colour**, **Number**, and **Name** options represent the mapping of the layers of the isometric drawing to the content for each layer.

• TIPS

- With the isometric drawing open in SmartSketch Drawing Editor, you can click Tools > Display Manager to see a list of the layers for the isometric drawing.
- Click **Hide** to close the grid.
- 5. Use **Default Color** to specify a default color to use for all layers if a color is not explicitly set.
- 6. Expand the **Definitions** group, and then click **Show** to open the **Definitions** data grid.

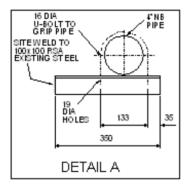
- 7. Define the layers on the isometric drawing as needed. For example, you can specify scale and line thickness.
- 8. Click Save to update the active style file with the new property settings.
 - \star IMPORTANT You must have write permission to the reference data to save changes to an isometric style.
- 9. Use the **Drawing Preview** panel to review your changes.
- NOTE For more information on the drawing layer and drawing definition options, see *Layers* and *Definitions* in the Smart 3D *Isometric Drawing Options Reference Guide*, which is available using the **Help > Printable Guides** command in the software.

Detail Sketches in Drawings

Detail sketches are small inserted drawings that provide additional details for components. The sketches are typically used to show more information about hangers and supports, branch connections, support lugs, and special welds.

★ IMPORTANT This functionality is applicable only to piping isometric drawings.

To include detail sketches on your drawing, you must prepare symbols in a graphics package such as SmartSketch, AutoCAD, or MicroStation. Then, you can specify options that control various characteristics of the detail sketch, such as the text formatting and placement. You also must map the sketches to part class names of components. If a component belonging to a specified part class is in the drawing, the software prints a callout next to the component, and the sketch is included on the drawing. An example detail sketch follows.



The software delivers example detail sketch templates to each client in the [Product Folder]\Drawings\Templates\DetailSketches folder. These templates provide an easy way to set up detail sketches for testing and review. The sketches are the proper size with correct positioning of labels, so you can save time by re-using these templates when creating your own.

Informational notes are similar to detail sketches. These notes refer to pipelines, spools, or components in the drawing. An example informational note follows.



Add a detail sketch

- 1. Open SmartSketch.
- 2. Create a symbol file.
 - TIP Alternatively, open one of the delivered symbol files (.sym format) and modify it. The delivered symbol files are located on each client in the [Product Folder]\Drawings\Templates\DetailSketches folder. Double-clicking these symbol files opens them in the SmartSketch. You can click the commands on the Help menu to find more information about creating and modifying symbols.
- 3. Save the symbol file.
- 4. Open the application and switch to the Drawings and Reports task.

5. Right-click an Isogen Isometric component or package in the **Management Console** or **Drawing Console**, and then select **Edit Options**.

Isogen Configuration opens.

- 6. On the Home screen, select **Detail Sketches** in the **Drawing Area** category pane.
 - The software switches to the **Configuration** view and displays the contents of the **Detail Sketches** panel.
- 7. Expand the **Settings** options group heading, and set **Path** to the location of the symbol file. For example, set the path to [*Product Folder*]\Drawings\Templates\DetailSketches\. You must add the \character on the end manually.
- 8. Set other options such as the file format, label type, and sketch position.
- 9. Return to the Home screen, and then select **Content** in the **Smart3D** pane.

 The software switches to the **Configuration** view and displays the contents of the **Content** panel.
- 10. Expand the **Detail Sketches** options group heading.
- 11. Select **Show Detail Sketch**, and then click **Show** to open the **Sketch Mapping** collection data grid.
- 12. Map the part classes and symbols.
 - TIP You can find a list of part classes in the catalog by switching to the **Catalog** task and browsing the hierarchy to **Drawings > Symbol Map**.
- 13. Click to update the active style file with the new property settings.
 - ★ IMPORTANT You must have write permission to the reference data to save changes to an isometric style.

■ NOTES

- The software delivers example detail sketch templates in .sym format to each client in the [Product Folder]\Drawings\Templates\DetailSketches folder. These templates provide an easy way to set up detail sketches for testing and review. The sketches are the proper size with correct positioning of labels, so you can save time by re-using these templates when creating your own.
- You also can create the sketch symbol in a graphics package with the capability of exporting to .dxf. Then, open the symbol in the SmartSketch to create the symbol file.

Custom Symbols for Isometric Drawings

You can easily create customized symbols for your isometric drawings using Symbol Editor. With this software, you can easily create new or modify existing XML-based or ASCII symbol files. The symbol editor software allows you to define and modify a component and then export the symbol to an Isogen ASCII file so that it can be used in the isometric drawings that you generate with the software. For information about downloading and installing Symbol Editor, please contact *Intergraph Support* (http://www.intergraph.com/support).

■ NOTES

- You must use Isogen Configuration to modify the style settings so that the software supports customized symbols that are in ASCII format.
- After creating the symbol file, you associate the file with an isometric style and test it by extracting isometric drawings.

What do you want to do?

- Configure the software to use an ASCII symbol file (on page 219)
- Create a custom symbol for isometric drawings (on page 220)

Configure the software to use an ASCII symbol file

EXAMPLE To use these customized symbols in Smart 3D, you must first use the Symbol Editor **Export ISOGEN Symbols** command to generate an Isogen ASCII (.asc) symbol file. For more information, see the *Symbol Editor User's Guide*, which is delivered with the Symbol Editor software.

- 1. Open the Drawings and Reports task.
 - Right-click an Isogen Isometric Drawing component or package in the **Management Console**, and then select **Edit Options**.
 - **Isogen Configuration** opens.
- 2. On the Home screen, select **Input Files** in the **Drawing Manager** category pane.
 - The software switches to the **Configuration** view and displays the contents of the **Input Files** panel.
- 3. Expand the **Summary** options group heading, and then click **Show** to open the **Input Files** grid
- 4. Select **ASCII-SYMBOLS** in the **Type** list.

- 5. Set the path to point to the appropriate .asc file in the **Path** box.
- 6. Click to update the active style file with the new property settings.
 - ★ IMPORTANT You must have write permission to the reference data to save changes to an isometric style.
- 7. Use the **Drawing Preview** panel to review your changes.

Create a custom symbol for isometric drawings

- 1. Open the symbol editor software.
- 2. Right-click a component type in the **Library Explorer**, and then select **New Symbol**.
 - **NOTE** Alternatively, you can a select a component type in the **Library Explorer**, and then click **Symbol > New Symbol**.
- 3. In the **New Symbol** dialog box, specify options for the new symbol.

TIPS

- Use the Original SKEY list to specify the standard symbol on which the new symbol is to be based.
- Select the Copy Symbol from box and keep the Library option selected.
- Use the Spindle SKEY list to specify the required spindle. For example, select the 01SP spindle.
- 4. Click OK.
- 5. Modify the symbol as necessary.

TIPS

- Use the commands on the **Symbol** menu to modify the symbol shape or add tapping points.
- Click in the Edit window and use the commands on the toolbar and the Symbol menu to draw additional lines for the symbol. Click Undo to correct any mistakes. Right-click to stop drawing lines.
- 6. Click File > Export ISOGEN Symbols, and save the symbol file in .asc format.
 - **NOTE** ASCII (.asc) format is recommended for the creation and maintenance of all Alias symbols. Binary (.bin) format symbol files may not be portable to future software releases.
- 7. Click **File > Exit** to close the editor.
- 8. Open Smart 3D, and switch to the Drawings and Reports task.
- Right-click an Isogen Isometric Drawing component or package, and then select Edit Options.

Isogen Configuration opens.

- 10. On the Home screen, select **Input Files** in the **Drawing Manager** category pane.

 The software switches to the **Configuration** view and displays the contents of the **Input Files** panel.
- 11. Expand the **Summary** group, and then click **Show** to open the **Input Files** data grid
- 12. Select ASCII-SYMBOLS in the Type list.
- 13. Click the ellipsis button in the **Path** box, and then browse to the symbols library file (.asc file) you created. If necessary, select **All Files (*.*)** in the **Files of type** field.
- 14. Click some isometric drawings to test.

Change Management in Piping Isometric Drawings

Change Management on an Isogen isometric drawing refers to the fact that the same information needs to display on the same sheet of an isometric drawing each time you extract the drawing. Change Management allows you to keep the overall drawing consistent in two ways:

- It maintains the same information per drawing sheet whenever possible.
- It prevents impact to other drawing sheets.

For example, a valve should not move from *Sheet 1* to *Sheet 2* in a piping isometric drawing simply because you added an olet to a pipe output to that same sheet. If the valve originally displayed on *Sheet 1*, it should always display on *Sheet 1*. However, if you add a gate valve and the MTO on *Sheet 1* is full, the software adds a new sheet called *Sheet 1A*. This way the gate valve is still on *Sheet 1*, but the MTO is not overcrowded.

Also, when Change Management is enabled, you should not modify the part numbers on the isometric drawing between updates. For example, assume that you placed a gate valve and updated the isometric drawing with Change Management enabled. The part number in the MTO might be 3 for the gate valve. Now, if you insert another gate valve on the pipeline and update the isometric drawing again, the old gate valve still has a part number of 3 and the new inserted gate valve has a part number of 4. This also applies to label callouts on the isometric drawing itself.

The Change Management functionality is enabled when you select **Change Management Enabled** in **Smart 3D > Drawing Creation > Overview**.

★ IMPORTANT This functionality is applicable only to piping isometric drawings.

■ NOTE You can override the Smart3D > Drawing Creation > Overview > Change Management Enabled option in the Isometric style by changing the drawing document Change Management property setting. For more information, see *Style Tab (Properties Dialog Box)* (on page 55).

Spool Number Consistency

Spool data is generated in the Piping task and stored in the model database. Therefore, spool numbers remain constant even if you re-extract a drawing.

NOTE If the **In Situ** spooling is active, the spool numbers could change depending on the types of changes that occurred in the model and/or settings in the spooling rules.

Weld Number Consistency

Weld number data is generated in the Piping task and stored in the model database. Therefore, weld numbers should not change even if you re-extract the drawing.

Material Part Number Consistency

Material part numbers in an MTO are driven by a label that, by default, looks for the *SequenceID* property of the part. One of the main purposes of Change Management is to have consistent part numbers on the MTO. This also applies to label call-outs for the Part number. However, Change Management does not affect your ability to apply another label, for example, to a gate valve.

Component Data Continuity

Any piping parts, components, instruments, or engineered items displaying on a given sheet do not change. If you re-extract the drawing, the components remain on the same drawing sheet. The same is true for the parts inclusion in a material list.

Weld Data Continuity

Welds display on a specific sheet on a piping isometric drawing. If you re-extract the drawing, the welds do not change sheets. The same is true for the weld list.

Spool Representation Continuity

Spools displaying on a given drawing sheet do not change when you re-extract the drawing unless a new spool is added or an existing spool is deleted. The software handles additions and deletions implicitly by the material (parts and components) managed on the drawing; however, the software also maintains spool integrity. For example, if one part is moved to another drawing sheet and the spool displayed is active, the entire spool moves with the part.

Material List Continuity

Each drawing sheet has its own material list. The items on the list correspond to those shown in the drawing. When you re-extract a drawing, the material list for any given component remains constant. For example, the schematic drawing on which a given commodity displays and the material list that includes that commodity always display on the same drawing sheet.

User-Defined Isometric Break Points

If you select **Smart3D > Drawing Creation > Overview > Change Management Enabled**, the software ignores all user-defined isometric break points. Only the sheet breaks from a previous extraction of the drawing is used for change management. If isometric break points are added or deleted from the model after change management is enabled, there is no impact on revised isometric drawings.

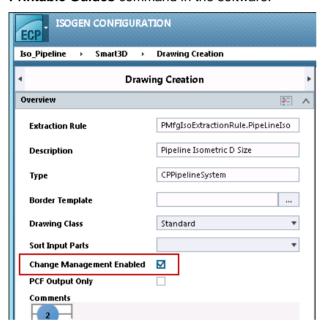
If change management is turned off (the option is not selected), the software honors the isometric break points.

Use piping isometric change management

The following procedures show how you must set your isometric style options in order to use change management for piping isometric drawings. Right-click a Piping Isometric Drawing component or package in the **Management Console** or **Drawing Console**, and then select **Edit Options** to open **Isogen Configuration**.

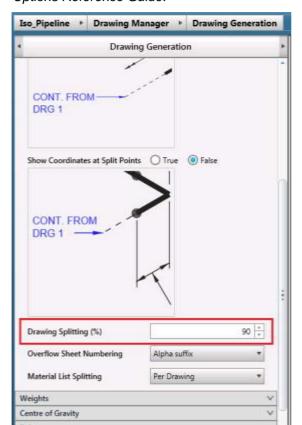
Set the Change Management Enabled Option

To turn on change management for piping isometric drawings, select **Smart 3D > Drawing Creation > Overview > Change Management Enabled**. For more information, see *Drawing Creation* in the *Isometric Drawing Options Reference Guide*, which is available with the **Help > Printable Guides** command in the software.



Set the Pipeline Splitting Option

When using change management for piping isometric drawings, set the **Drawing Manager** > **Drawing Generation** > **Drawing Splitting** > **Drawing Splitting** (%) n to at least 90 (the default

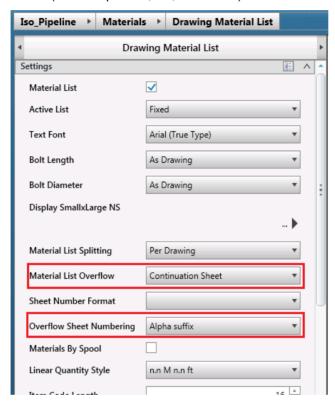


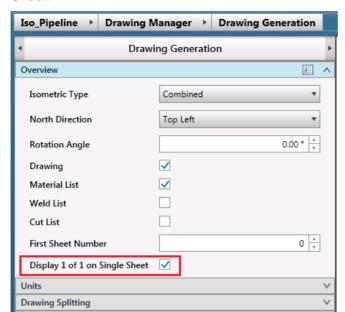
value). For more information on this option, see *Drawing Splitting* in the *Isometric Drawing Options Reference Guide*.

Set the Material List Options

To handle material list overflows from sheet to sheet without breaking change management, we recommend the following isometric option settings:

- Set Materials > Drawing Material List > Settings > Material List Overflow to Continuation Sheet.
- 2. Set Materials > Drawing Material List > Settings > Overflow Sheet Numbering to Alpha suffix (for example: 2A, 3A, and so on).





3. Select Drawing Manager > Drawing Generation > Overview > Display 1 of 1 on Single Sheet.

For more information on setting material list options, see Settings in the Smart 3D Isometric Drawing Options Reference Guide.

See Also

Edit Options (Isogen Isometric Drawing Component Shortcut Menu) (on page 134)

SECTION 19

Drawing Configuration Tools

The tools listed below are also included with Smart 3D. When used, these tools can further streamline the process for configuring drawing output.

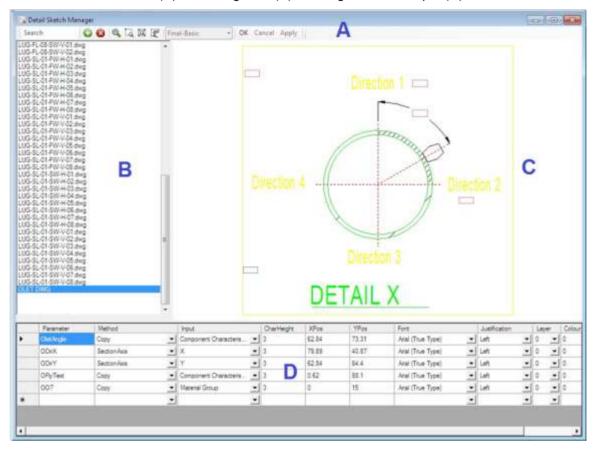
- Detail Sketch Manager helps you configure detail sketches for isometric drawing output. The Detail Sketch Manager allows you to place parameters on the selected detail sketch so that the correct relevant values, such as weld numbers, part numbers, and angles, are plotted on the isometric drawing. For more information, see Detail Sketch Manager (on page 229).
- Drawing Setup Tool includes a limited set of drawing options that can assist you in configuring the drawing template. The drawing template is usually a company standard backing sheet on which Isogen plots the drawing and related data such as the material take-off, welding report, and project-related data. The Drawing Setup Tool has a graphical interface, overlaying the location of key items onto the backing sheet. For more information, see Drawing Setup Tool (on page 234).

In This Section

Detail Sketch Manager	229
Drawing Setup Tool	234

Detail Sketch Manager

To open the **Detail Sketch Manager**, navigate to **Drawing Area > Detail Sketches > Settings**, and then click **Library**. The **Detail Sketch Manager** window consists of a toolbar (**A**), a list of available detail sketches (**B**), a viewing area (**C**), and a grid for data input (**D**).



A - Detail Sketch Manager Toolbar

Search

Filters the list of available detail sketches based on user-defined criteria. As you type text into the **Search** box, the feature responds with an implied wild card at the beginning and at the end of the character string. For example, if you type **B**, the list of available sketches updates to display only those sketches with a **B** in the filename based on the defined criteria, ***B***.

Add

Inserts a blank row at the bottom of the grid. You can also insert a row by clicking in the * row at the bottom of the grid.

Delete

Removes the currently selected row from the grid, as well as the detail sketch identifier, or that the data defined.



Increases or decreases the display size of detail sketch objects in the viewing area. You can zoom in to get a closer view of an object, or zoom out to view more of the detail sketch at a reduced size. Click the left mouse button and drag the pointer upward in the window to increase the view of an object as though you were moving closer to it. Drag the pointer downward in the window to reduce the view as though you were moving further away from the object.

Zoom Area

Increases the view magnification of an area in the detail sketch that you define with two points.



Zoom All

Displays the entire detail sketch in the viewing area.



Repositions the detail sketch in the viewing area so that you can view another section of the sketch without changing the view magnification.

Style

Displays the active style. This option is read only.

OK

Prompts you to save any unsaved data. Click Yes to save the data in the grid to the current style and close the **Detail Sketch Manager**.

Cancel

Discards any unsaved data in the grid, and closes the **Detail Sketch Manager**.

Apply

Saves the data that has been defined in the grid for the current sketch, but leaves the **Detail Sketch Manager** open so that you can define parameters for another sketch.

B - Detail Sketch List

Displays the detail sketches that are available for the active style. When you select a detail sketch in the list, a graphical view of the sketch displays in the detail sketch viewing area, as shown below.

■ NOTES

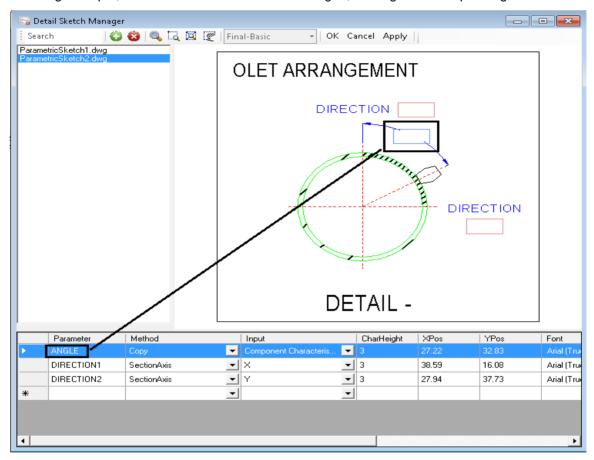
- The current version of the software does not support DGN V8 CELL libraries. All other file formats, DWG, DGN (CELL Library), and SYM (IGR/SHA), are fully supported.
- You can use the Search box on the Detail Sketch Browser toolbar to limit the list of available sketches.

C - Detail Sketch Viewing Area

Displays a graphical view of the selected detail sketch.

D - Grid Area

Allows you to position parameters directly onto the detail sketch. Parameters are derived from the component on which the detail sketch is placed. Each line of data in the grid displays a box at the selected X,Y location on the detail sketch. When you select a row in the grid, the corresponding detail sketch parameter box is considered active and displays as blue in the detail sketch viewing area. An inactive detail sketch parameter box displays as red. In the following example, the **ANGLE** row is selected in the grid, making the corresponding box active.



Parameter

Specifies the user-supplied detail sketch identifier.

Method

Specifies how the required data is to be extracted from the piping object data (POD) file for isometric drawing output. The following table lists the supported methods, along with any required input parameters.

Selected Method	Required Input Parameter	Description
Сору	Enter any valid attribute or property of the component, its parent pipeline, or its material.	Returns the value of the specified property. This can include the part number, the weld number, or any other known property.
SectionAxis	X	Returns the X- or Y-axis label. The Y-axis label for a section through the component, based on the component HeaderOrientation property is as follows: Vertical returns E (or as modified by ATEXT -248) Horizontal returns U (-244) Sloped/falling returns U (-244) Non-orthogonal returns blank The X-axis label for a section through the component, based on the component HeaderOrientation property is as follows: Vertical returns S (-247) Horizontal returns orthogonal vector to header and Up, E for N/S header, S for E/W header, compound direction otherwise. Sloped/falling returns same as Horizontal Non-orthogonal returns blank
SectionAngleFromClosestAxis	Specify a component property, such as BranchAngle , or attribute that contains an angle. For example, you can enter C.BranchAngle or C.COMPONENT-ATTRIBUTE1 .	Converts the input angle, which is 0-360, to an angle based on the closest axis. Angle = 0 to 45 returns angle Angle = 45.1 to 89.9 returns (90 – angle) Angle = 90 to 135 returns (angle – 90) Angle = 135.1 to 179.9 returns (180 – angle) Angle = 180 to 225 returns (angle – 180) Angle = 225.1 to 269 returns (270 – angle) Angle = 270 to 315 returns (angle – 270) Angle = 315.1 to 359.9 returns (360 – angle) Invalid input (not a number in the range 0-360) returns zero (0)
MainWeldNumber	N/A	Returns the REPEAT-WELD-IDENTIFIER of the main (header) weld of an olet.

Selected Method	Required Input Parameter	Description
BranchWeldNumber	N/A	Returns the REPEAT-WELD-IDENTIFIER of the branch weld of an olet, set-on or tee.
SupportWeldNumber	Enter 1-n.	Returns the REPEAT-WELD-IDENTIFER of the nth support weld.
LugPartNumber	Enter 1-n.	Returns the REPEAT-PART-NUMBER of the nth lug
AdditionalMaterialPartNumber	Enter 1-n.	Returns the REPEAT-PART-NUMBER of the nth additional material

Input

Defines the parameter required by the selected **Method**. Select a parameter from the list, or type the parameter directly into the box. For more information about methods and their associated input parameters, refer to the above table.

CharHeight

Defines the height of the character on the isometric drawing. It also sets the size of the box used to place text. The default setting is 3.

XPos

Defines the X-position of the detail sketch identifier relative to the bottom left corner of the detail sketch.

YPos

Defines the Y-position of the detail sketch identifier relative to the bottom left corner of the detail sketch.

Font

Specifies the type of font used to display text.

Justification

Defines the justification for the text. You can select Left, Centre, or Right.

Layer

Specifies the layer of the CAD file to use.

Colour

Specifies the index number that represents a specific color as defined in the output drawing software.

Rotation

Defines the angle that text is allowed to be rotated in output.

Drawing Setup Tool

Customizes the content and format of the isometric drawing for the active isometric drawing style.

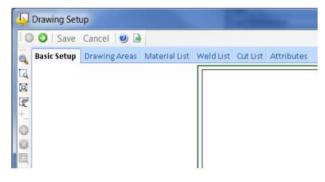
NOTE You can access the **Drawing Setup Tool** only from within **Isogen Configuration**. For more information, see *Open the Drawing Setup Tool* (on page 237).

Many of the settings that you can configure using **Isogen Configuration** can be time-consuming. The **Drawing Setup Tool** enables you to edit drawing style properties in a graphic view, which can often be much easier. You can graphically define the position of the drawing area on the backing sheet and specify which user-defined reports, if any, are output on the drawing. You can also use the tool to define the attributes that display on the drawing frame. The options that you define in the tool can then be saved to the current isometric drawing style so that they are applied to the generated drawings.

The **Drawing Setup Tool** consists of a series of tabbed pages, each of which contains a set of options and properties that are specific to a particular aspect of the drawing setup, and a horizontal and vertical toolbar. The pages listed below appear by default for each isometric drawing style:

- Basic Setup
- Drawing Areas
- Attributes

If a user-defined Material List, Weld List, or Cut List report is currently associated with the selected isometric drawing style, or if you choose to add one of those reports during the **Drawing Setup Tool** process, a tabbed page specific to that report type is automatically displayed so that you can configure those settings as well. The example below shows the complete set of tabbed pages that can display in the tool:



Basic Setup Page (on page 247)

Drawing Areas Page (on page 249)

Material List Page (on page 253)

Weld List Page (on page 259)

Cut List Page (on page 262)

Attributes Page (on page 265)

Add/Remove Attributes Dialog Box (on page 267)

The commands on the vertical and horizontal toolbars provide options for navigating through the tool and for manipulating the graphical view. For example, you can adjust the view to display the entire backing sheet, or you can focus on a specific area, such as the drawing frame. You can also increase and decrease the viewing area. Decreasing causes everything within the window to appear larger, while increasing causes everything within the window to appear smaller. Toolbar commands are also available that allow you to control the content of the user-specified reports and the attributes that are plotted in the title block.

Drawing Setup Tool Toolbar

Vertical Toolbar:



Zoom In/Out increases or decreases the display size of the drawing template. You can zoom in to get a closer view of an area or zoom out to view more of the drawing template at a reduced size. Click the left mouse button and drag the pointer upward in the window to increase the view of an object as though you were moving closer to it. Drag the pointer downward in the window to reduce the view as though you are moving further away from the object.

- Zoom Window increases the view magnification of an area in the drawing template that you define by creating a fence.
- **Zoom to Fit** displays the entire contents of the drawing template in the window.
- Pan repositions the drawing template in the display window so that you can view another section of the template without changing the view magnification.
- + Add / Remove Attributes opens the Add/Remove Attributes dialog box. This dialog box allows you to specify the attributes that are available in the **Attribute** list in the **Attributes Grid** window. The command is available only when the **Attributes** page is active.
- Add Column inserts a column to the selected report so that an additional attribute can be plotted in the report. This command is available only when the Material List, Weld List, or Cut List page is active.

- Delete Column removes the selected column of component attribute data from the plotted report. This command is available only when you select a Column category label on the Material List, Weld List, or Cut List page.
- Zoom to List Attributes zooms in on the selected attribute. Each time you click Zoom to List Attributes , the view zooms to the selected attribute in the title block of the drawing frame or on the reports.

Horizontal Toolbar:

- Previous Page returns to the previous page of the tool.
- Next Page proceeds to the next page of the tool.
- Welp opens the on-line documentation delivered with the software and displays information about the tool.

OK saves the options and parameters that you have defined.

Cancel undoes the options and parameters that you have defined.

Exit closes the tool. If you have not saved your changes, the software prompts you to do so.

■ NOTES

- The four graphic control commands on the vertical toolbar (Zoom In/Out , Zoom Window , Zoom to Fit ⋈, and Pan) can be used any time while running the tool. The availability of the remaining vertical toolbar commands is determined by the active page.
- You can navigate back and forth sequentially through the pages using Previous Page and Next Page on the horizontal toolbar. Alternatively, click the tab to go directly to a specific page.

What do you want to do?

- Open the Drawing Setup Tool (on page 237)
- Specify the backing sheet (on page 237)
- Define drawing margins and report layout (on page 237)
- Customize report settings (on page 240)
- Define drawing frame attributes (on page 245)
- Update the style (on page 247)

Open the Drawing Setup Tool

1. Right-click an Isogen Isometric Drawing component or package in the **Management Console** or **Drawing Console**, and then select **Edit Options**.

The software opens Isogen Configuration.

- 2. Click **Drawing Setup Tool** in the bottom right corner of the **Isogen Configuration** window. *The software opens the Drawing Setup Tool.*
- 3. Use the options on the tabbed pages to customize the isometric drawing output as needed.
- 4. When finished, click settings.

Specify the backing sheet

- 1. Open the Drawing Setup Tool (on page 237).
- 2. On the **Basic Setup** page, click **Browse**, and navigate to the backing sheet template file that you want to use.
 - **NOTE** The delivered backing sheets are stored as .igr files in the [Reference Data Product Folder]\SharedContent\PmfgIsoStyleData folder on the server.
- 3. Set the required drawing size. If you select **Custom**, you must specify values for **Width** and **Height**.
- 4. Under **Reports**, specify the reports that you want Isogen to plot on the isometric drawing. If a user-defined material list is to be plotted, you must specify the required number of sections. By default, two sections are output on the material list.
- 5. Click Save to update the active isometric style with the new settings.
- **NOTE** If you change the backing sheet template file, the software copies the referenced file to the catalog database and accesses the file from the database rather than from the folder location specified in the **Template** box. As such, when the **Drawing Setup Tool** is reopened after the template file has been changed, the **Template** box displays a path to the Temp folder.

Define drawing margins and report layout

You can customize the left, right, top, and bottom margins of the drawing area, either interactively with the mouse or by typing new values in the **Drawing Areas** window. Likewise, you can use either method to set the start positions of where Isogen plots the material, weld, and cut lists on the isometric drawing.

Set Drawing Margins Graphically

- 1. Open the Drawing Setup Tool (on page 237).
- 2. On the **Drawing Areas** page, place the cursor on the grab handle for the margin you want to change.

The cursor updates from a pointer to a two-headed arrow cross, indicating the direction in which you can drag the boundary.



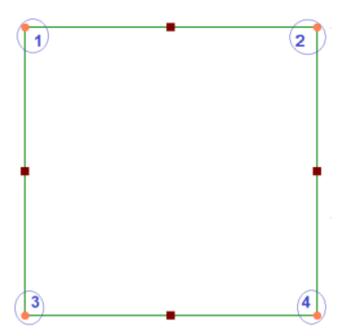
- TIP Alternatively, place the cursor on one of the boundaries. The cursor updates from a pointer to a cross-hair, indicating that the margin is ready to be resized.
- 3. Drag the boundary to the new position, and then release the mouse.

As you drag the boundary, the corresponding margin value in the **Drawing Areas** window dynamically updates.

TIP To accurately fit the drawing area, zoom in to each corner of the area being defined.

NOTE Using the corner grab handles, you can move two margins simultaneously while the other two margins remain unchanged.

- Grab handle 1: Resizes the left and top margins.
- Grab handle 2: Resizes the right and top margins.
- Grab handle 3: Resizes the left and bottom margins
- Grab handle 4: Resizes the right and bottom margins.



Set Drawing Margins Using the Grid

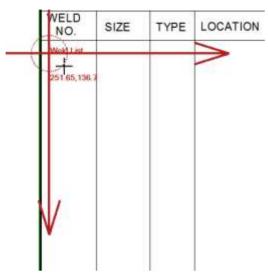
- 1. Expand **Drawing Area Margins** in the **Drawing Areas** window, and click the margin you want to resize.
- 2. Type the new value.

The drawing boundary box in the View window updates accordingly.

Set Report Start Position Graphically

1. On the **Drawing Areas** page, place the cursor on the appropriate report anchor.

The cursor updates from a pointer to a cross-hair, indicating that the anchor is ready to be moved.



NOTE In the above example, the numbers underneath the cross-hairs indicate the X and Y coordinate values.

2. Move the anchor to the required location, and then release the mouse.

As you drag the anchor, the corresponding **X** and **Y** values that display with the anchor and in the **Drawing Areas** window dynamically update.

Set Report Data Start Position Using the Grid

- 1. Expand the appropriate list category in the **Drawing Areas** window, click **X**, and type the new value.
- 2. Click Y, and type the new value.

The drawing boundary box in the **View** window updates accordingly.

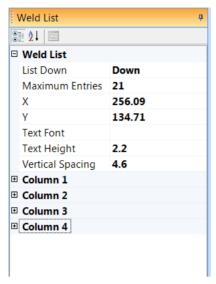
Customize report settings

You can use the grid to define formatting properties such as text font, height, and the direction in which the report data is plotted. Additionally, some of the properties that control the layout of the report data can be customized graphically using the mouse.

Set Up the Report Using the Grid

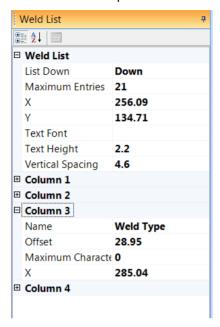
- 1. *Open the* **Drawing Setup Tool** (on page 237), and select the tab that corresponds to the report you want to customize.
 - **NOTE** Use the **Reports** section on the **Basic Setup** page to add and remove the reports to be plotted on the isometric drawing.
- 2. If needed, click **Zoom to List Attributes** on the vertical toolbar to zoom in on the report section of the backing frame.

3. Expand the report category node to expose those properties. In the example below, the **Weld List** category node is expanded.



- 4. Use List Down to specify the direction in which Isogen is to plot the report data.
- 5. Click **Maximum Entries**, and set the maximum number of entries allowed in the report before an overflow sheet is generated.
- 6. Type the appropriate **X** and **Y** values to set the horizontal and vertical start positions of the first line of the report.
 - **NOTE** Alternatively, you can define these X and Y settings graphically. For more information, see *Define X,Y, and Offset Settings Graphically* below.
- 7. Select the font to be used to display the report data in the **Text Font** list.
- 8. If needed, modify the values for **Text Height** and **Vertical Spacing**.

9. Expand the column nodes to display those properties. In the example below, the **Column 3** node has been expanded.



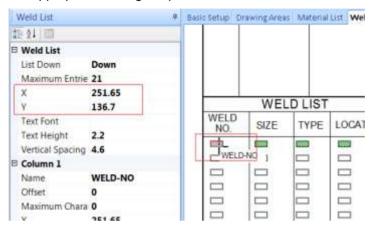
- 10. Modify the component attribute settings as needed.
 - Use the Name list to specify which component attribute is to be plotted.
 - Use the Offset and X boxes to set the plot start position for the component attribute data. These settings can also be defined graphically. For more information, see Define X, Y, and Offset Settings Graphically below.

Define X,Y, and Offset Settings Graphically

Modifying the report layout graphically involves using the mouse to position report data. Using the mouse, you can define the horizontal and vertical position of the first line of data that is plotted on the report. You can also define an offset from the start position that the other columns of data use.

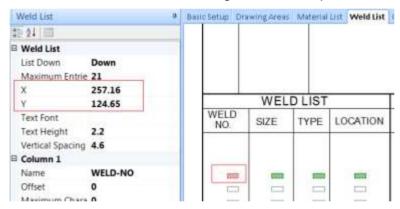
Define the report origin graphically

- 1. Place the mouse on the red rectangle in the first column of the report block.
- 2. When the mouse changes to a cross-hairs, as shown in the example below, drag the box in the appropriate left, right, up, or down direction.



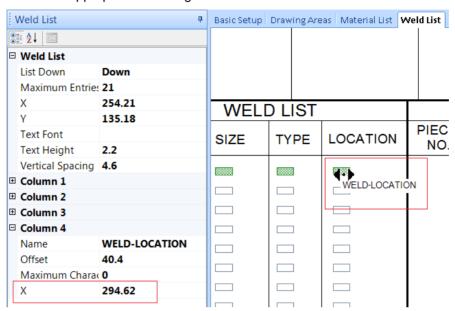
The **X** and **Y** values dynamically update as you drag the rectangle.

3. Release the mouse when the rectangle has been re-positioned as needed.



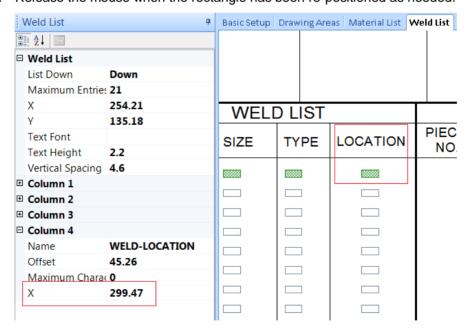
Define an offset from the report origin graphically

- 1. Place the mouse on the green rectangle in the column of data that you want to reposition.
- 2. When the mouse changes to a two-side arrow, as shown in the example below, drag the box in the appropriate left or right direction.



The **Offset** and **X** values dynamically update as you drag the rectangle.

3. Release the mouse when the rectangle has been re-positioned as needed.



Add and Remove Report Columns

1. Click **Add Column** on the vertical toolbar.

The software adds a node to the tree view, and adds a column of empty rectangles to the appropriate report location in the **View** window.

NOTE Each empty rectangle represents an attribute value. To view the empty rectangles, set **Maximum Entries** and **Vertical Spacing** to a value greater than zero (0).

- 2. Define settings for Name, Offset, Maximum Characters, and X as needed.
- 3. To remove a column from the report, select the column in the tree view, and then click **Delete Column** on the vertical toolbar.

The software removes the column heading from the tree view and the column of rectangles from the report location.

Define drawing frame attributes

Add or Remove a Drawing Frame Attribute

- 1. Open the **Drawing Setup Tool (on page 237),** and select the **Attributes** tab. The **Attributes** page displays.
- 2. Click **Add/Remove Attributes** + on the vertical toolbar.

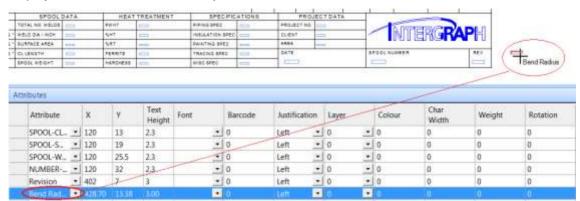
The software opens the Add/Remove Attributes dialog box.

- 3. In the Available list, select the attribute to add.
 - TIP Hold down the CTRL key to select multiple attributes.
- 4. Click Add to add the attribute In Use list.
- 5. Click **OK** to close the dialog box.

The software inserts the attribute at the bottom of the **Attributes Grid** window, and places an empty rectangle on the drawing frame in the vicinity of the last attribute placed on the grid.

Position the rectangle as required, by graphically selecting and moving it, and then adjust its position by editing it in the **Attributes Grid** window.

NOTE When you select a rectangle in the drawing frame, its outline color changes to red and the corresponding row in the **Attributes Grid** window is highlighted. The cursor changes to a cross-hair, indicating that it can be moved. Finally, the software displays a tooltip showing the display name as defined in the Project defaults.



Remove a Drawing Frame Attribute

1. Select the Attributes tab in the Drawing Setup Tool.

The Attributes page displays.

2. Click **Add/Remove Attributes** + on the vertical toolbar.

The software opens the Add/Remove Attributes dialog box.

- 3. In the **In Use** list, select the attribute to remove.
- 4. Click **Remove Attribute** o to remove the attribute from **In Use** list.
- 5. Click **OK** to close the dialog box.
- 6. The software removes the row containing the attribute data from the **Attributes Grid** window, removes the corresponding rectangle from the drawing frame.

Graphically Move a Drawing Frame Attribute

1. Click once on the attribute to move.

The software highlights the attribute in red on the drawing frame, and the pointer changes to a cross.

- 2. Drag the attribute to its new position, and release the mouse.
- 3. NOTE As you drag the attribute, the corresponding **X** and **Y** settings in the **Attributes**Grid window dynamically update.

Modify a Drawing Frame Attribute

1. Click an attribute rectangle in the drawing frame.

The software highlights the attribute in the **Attributes Grid** window.

2. Modify the attribute values as needed in the **Attributes Grid** window.

Update the style

In order for the changes that you make in the **Drawing Setup Tool** to be effective, you must click save in the bottom right corner of the ISOGEN Configuration window to update the active isometric drawing style file with the new settings.

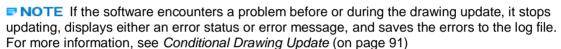
View the results

When you have completed using the **Drawing Setup Tool**, it is recommended that you review how your changes affect the isometric drawing output. In that way, if anything appears incorrect, you can re-open the tool and correct the problems.

•

- Right-click a document.
- 2. On the shortcut menu, select **Update Now** to update the drawing.

The icon for the out-of-date document changes to show that it is updated \checkmark .



- 1. Right-click a document.
- 2. On the shortcut menu, select **Update Now** to be updated locally.

The icon for the out-of-date document changes to show that it is updated \checkmark .



■ NOTE If the software encounters a problem before or during the drawing update, it stops updating, displays either an error status or error message, and saves the errors to the log file. For more information, see *Conditional Drawing Update* (on page 91)

Basic Setup Page

Controls the drawing format, size, and the template file used as the backing sheet. You can also specify which user-defined reports are to be plotted on the isometric drawing when it is generated.

Template

Displays the file to be used for the drawing backing sheet. Click **Browse** ____, and navigate to the required template file.

Drawing Output Format

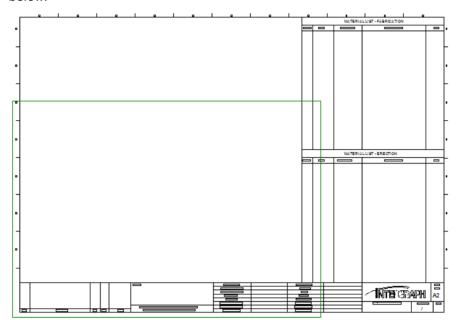
This option is not available in Smart 3D. The only supported output file format is Shape2d (SHA).

Allow ISOGEN to generate standard drawing frame

This option is not available in the current version of the software.

Drawing Size

Displays the standard paper size for the isometric drawing. Select **A0**, **A1**, **A2**, **A3**, **A4**, **ANSI** "A", **ANSI** "B", **ANSI** "C", **ANSI** "D", **ANSI** "E", or **Custom**. The drawing size that you select is shown superimposed in green on the backing sheet, as shown in the example below.



Width

Sets the width of the drawing. This option is available only if you set **Drawing Size** to **Custom**.

Height

Sets the height of the drawing area. This option is available only if you set **Drawing Size** to **Custom**.

Backing Sheet Units

This option is not available in the current version of the software.

User Defined Material List

Indicates whether a Material List report is output on the isometric drawing. To include this report on the isometric drawing, select its check box. If you select this option, you must also specify the **Number of sections**. Clear the check box to suppress the report output.

User Defined Weld List

Indicates whether a Weld List report is output on the isometric drawing. To include this report on the isometric drawing, select its check box. Clear the check box to suppress the report output.

User Defined Cut List

Indicates whether a Cut List report is output on the isometric drawing. To include this report on the isometric drawing, select its check box. Clear the check box to suppress the report output.

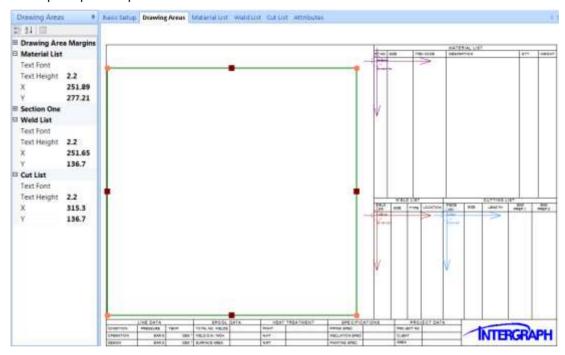
Number of sections

Specifies the number of sections included for the user-defined Material List report that is output on the isometric drawing. Select **One**, **Two**, or **Three**. This option is available only if **User Defined Material List** is selected.

Drawing Areas Page

Defines the drawing area, which includes the drawing sheet and the areas of the drawing frame where Isogen plots the user-defined reports. The **Drawing Areas** page of the tool is divided into two sections.

- Drawing Areas window displays a grid view of drawing and report layout properties. In the View window, a green box with grab handles identifies the drawing boundary. In the report area, a colored anchor indicates the start position of the report data.
- View window displays the selected style's default backing sheet with the drawing area superimposed upon it.



In the **View** window, you can graphically customize the left, right, top, and bottom margins of the drawing and define the layout and appearance of the report data that is plotted on the isometric drawing. You can also type new values in the **Drawing Areas** window. For more information, see *Define drawing margins and report layout* (on page 237).

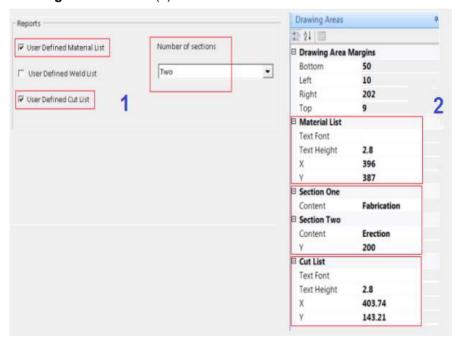
Drawing Areas Window

Any of the categories and related properties listed below can appear on the **Drawing Areas** window, depending on the settings that have been defined on the **Basic Setup** page.

■ NOTE The **Drawing Areas** window can be displayed, hidden, docked, maximized, tabbed, or made to float. For more information, see *Moving and Resizing Windows* in *Drawing Setup Tool* (on page 234).

Categorized

Groups drawing properties in the **Drawing Areas** window by category. At a minimum, the panel displays the **Drawing Area Margins** category, which contains margin settings for the drawing area. Additional categories, such as **Material List**, **Cut List**, and **Weld List**, display only if the corresponding report option is selected on the **Basic Setup** page. These categories contain format and layout properties specific to the data that is plotted on a report. Another set of categories can also display on the window: **Section One**, **Section Two**, and **Section Three**. These categories, which contain properties that are specific to the different sections of the material list, are controlled by the **Number of sections** setting on the **Basic Setup** page. The example below illustrates the relationship between the reports options selected on the **Basic Setup** page (1) and the categories that appear on the **Drawing Areas** window (2).



Groups drawing properties in the Drawing Areas window as a flat, alphabetical list.

Drawing Area Margins

Groups the properties that define the drawing margins. Margins are measured relative to the drawing sheet. For more information, see *Define drawing margins and report layout* (on page 237).

- Bottom sets the distance between the outer edge of the drawing sheet and the outer line of the drawing frame at the bottom of the drawing. Type a value (real number) in mm or inches, depending on the Backing Sheet Units setting on the Basic Setup page.
- Left sets the distance between the outer edge of the drawing sheet and the outer line of the drawing frame on the left side of the drawing. Type a value (real number) in mm or inches, depending on the **Backing Sheet Units** setting on the **Basic Setup** page.
- Right sets the distance between the outer edge of the drawing sheet and the outer line
 of the drawing frame on the right side of the drawing. Type a value (real number) in mm
 or inches, depending on the Backing Sheet Units setting on the Basic Setup page.
- **Top** sets the distance between the outer edge of the drawing sheet and the outer line of the drawing frame at the top of the drawing. Type a value (real number) in mm or inches, depending on the **Backing Sheet Units** setting on the **Basic Setup** page.

Material List

Groups layout and format properties for material list report data on the isometric drawing. This collection of properties appears only if **User Defined Material List** is selected on the **Basic Setup** page.

- Text Font specifies the font used to display material list report data. Select the required font in the Text Font list.
- Text Height specifies the height of the text that appears in the material list report. This
 setting applies to all sections of the material list. Type a value (real number) in mm or
 inches, depending on the Backing Sheet Units setting on the Basic Setup page.
- **X** sets the bottom left X, or horizontal, position of the first line of the material list plotted on the isometric drawing. Type a value (integer).
- Y sets the bottom left Y, or vertical, position of the first line of the material list plotted on the isometric drawing. Type a value (integer).

Weld List

Groups layout and format properties for weld list report data on the isometric drawing. This collection of properties is shown only if **User Defined Weld List** is selected on the **Basic Setup** page.

- Text Font specifies the font used to display weld list report data. Select the required font in the Text Font list.
- Text Height specifies the height of the text that appears in the weld list report. Type a
 value (real number) in mm or inches, depending on the Backing Sheet Units setting on
 the Basic Setup page.
- **X** sets the bottom left X, or horizontal, position of the first line of the weld list plotted on the isometric drawing. Type a value (integer).
- Y sets the bottom left Y, or vertical, position of the first line of the weld list plotted on the isometric drawing. Type a value (integer).

Cut List

Groups layout and format properties for cut list report data on the isometric drawing. This collection of properties appears only if **User Defined Cut List** is selected on the **Basic Setup** page.

- Text Font specifies the font used to display cut list report data. Select the required font
 in the list.
- Text Height specifies the height of the text that appears in the cut list report. Type a
 value (real number) in mm or inches, depending on the Backing Sheet Units setting on
 the Basic Setup page.
- **X** sets the bottom left X, or horizontal, position of the first line of the cut list plotted on the isometric drawing. Type a value (integer).
- Y sets the bottom left Y, or vertical, position of the first line of the weld list plotted on the isometric drawing. Type a value (integer).

Section One

Groups layout and format properties for the report data contained in Section 1 of the material list. This category appears only if **User Defined Material List** is selected on the **Basic Setup** page.

 Content indicates the category of data plotted in Section 1 of the material list. Select Fabrication, Erection, Offshore, or Erection/Offshore. The default setting is Fabrication.

Section Two

Groups layout and format properties for the report data contained in Section 2 of the material list. This category appears only if **Number of sections** on the **Basic Setup** page is set to **Two** or **Three**.

- Content indicates the category of data plotted in Section 1 of the material list. Select Fabrication, Erection, Offshore, or Erection/Offshore. The category that you select must not be the same as the Content setting for Section 1. The default setting is Erection.
- Y sets the Y, or vertical, offset for Section 2 of the material list with reference to the X and Y positions of the material list. Type a value (integer) in mm or inches, depending on the **Backing Sheet Units** setting on the **Basic Setup** page.
- X specifies the X (or horizontal) offset for Section 2 of the material list with reference to the report origin defined by the X and Y settings under the Material List category. Type a value (integer) in mm or inches, depending on the Backing Sheet Units setting on the Basic Setup page.

Section Three

Groups layout and format properties for the report data contained in Section 3 of the material list. This category appears only if **Number of sections** on the **Basic Setup** page is set to **Three**.

- Content indicates the category of data plotted in Section 3 of the material list. Select Fabrication, Erection, Offshore, or Erection/Offshore. The category that you select must not be the same as the Content setting for Section 1 or Section 2. The default setting is Offshore.
- Y sets the Y, or vertical, offset for Section 3 of the material list with reference to the X and Y positions of the material list. Type a value (integer) in mm or inches, depending on the Backing Sheet Units setting on the Basic Setup page.
- X specifies the X (or horizontal) offset for Section 3 of the material list with reference to the report origin defined by the X and Y settings under the Material List category. Type a value (integer) in mm or inches, depending on the Backing Sheet Units setting on the Basic Setup page.

Material List Page

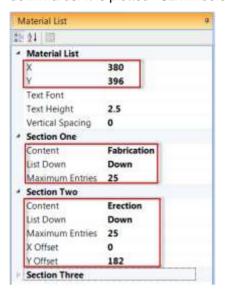
Defines the layout and appearance of the report data in the material list that is plotted on the isometric drawing. When Isogen processes the style to generate an isometric drawing, the material list is automatically populated to report the materials in the pipeline (or spool). The **Drawing Setup Tool** displays this page only when **User Defined Material List** is selected on the **Basic Setup** page. The **Material List** page is divided into two sections.

- Material List window displays properties that control the materials report set up, contents, and location (also shown graphically). These properties can be customized to meet your specific isometric drawing requirements. For more information, see *Customize report* settings (on page 240).
- View window displays a copy of the selected style's default backing sheet with the location of the materials report data superimposed upon it.

Material List Window

The categories and related properties listed below can appear on the **Material List** window, depending on the reports options that are selected on the **Basic Setup** page. The data in the example below defines a two section material list. The first section (**Section 1**), starting at 380mm x 396mm, contains a list of up to 25 Fabrication materials plotted downwards (**Down**).

The second section (**Section Two**) contains a similar number of Erection materials also plotted downwards. It is plotted 182mm below the first section.



NOTES

- Many of the properties listed below can also be defined or modified on the Basic Setup page of the tool.
- The Material List window can be displayed, hidden, docked, maximized, tabbed, or made to float. For more information, see Moving and Resizing Windows in Drawing Setup Tool (on page 234).

• Categorized

Groups material list properties in the **Material List** window by category. The **Number of sections** setting, which is defined on the **Basic Setup** page, determines whether the categories **Section Two** or **Section Three** are displayed. The *Section* categories contain properties that are specific to a particular section of the material list.

2↓ Alphabetical

Groups material list properties in the Material List window as a flat, alphabetical list.

Material List

Groups layout and format properties for material list report data, including the report origin and the text size and font.

- **X** sets the bottom left X, or horizontal, position of the first line of the material list that is plotted on the isometric drawing. Type a value (integer).
- Y sets the bottom left Y, or vertical, position of the first line of the material list that is plotted on the isometric drawing. Type a value (integer).
- Text Font specifies the font used to display material list report data. Select the required font in the Text Font list.
- Text Height specifies the height of the text that appears in the material list report. This
 setting applies to all sections of the material list. Type a value (real number) in mm or
 inches, depending on the Backing Sheet Units setting on the Basic Setup page.
- Vertical Spacing specifies the vertical spacing between the lines of data. The vertical spacing applies to all sections in the material list. Type a value (real number) in mm or inches, depending on the Backing Sheet Units setting on the Basic Setup page.

Section One

Groups layout and format properties for the report data contained in Section 1 of the material list.

- Content displays the category of data plotted in Section 1 of the material list. Select Fabrication, Erection, Offshore, or Erection/Offshore. The default setting is Fabrication.
- List Down controls the direction in which the material list is printed with reference to the X and Y settings under the Material List category. If you select Down, Isogen plots the material list from top to bottom. If you select Up, the material list is plotted from bottom going up.
- Maximum Entries sets the maximum number of entries in the material list before an overflow sheet is generated. Type the required value (positive integer).

Section Two

Groups layout and format properties for the report data contained in Section 2 of the material list. This category appears on the panel only if **Number of sections** on the **Basic Setup** page is set to **Two** or **Three**.

- Content displays the category of data plotted in Section 1 of the material list. Select Fabrication, Erection, Offshore, or Erection/Offshore. The category that you select must not be the same as the category defined for Section 1. The default setting is Erection.
- List Down controls the direction in which the material list is printed with reference to the X and Y settings under the Material List category. If you select Down, Isogen plots the material list from top to bottom. If you select Up, the material list is plotted from bottom going up.

- Maximum Entries sets the maximum number of entries in the material list before an overflow sheet is generated. Type the required value (positive integer).
- X Offset specifies the X (or horizontal) offset for Section 2 of the material list with reference to the report origin defined by the X and Y settings under the Material List category. Adding a Section 2 X offset results in a horizontal double banking of the BOM. Type a value (integer) in mm or inches, depending on the Backing Sheet Units setting on the Basic Setup page.
- Y Offset specifies the Y (or vertical) offset for Section 2 of the material list with reference to the report origin defined by the X and Y settings under the Material List category. Type a value (integer) in mm or inches, depending on the Backing Sheet Units setting on the Basic Setup page.

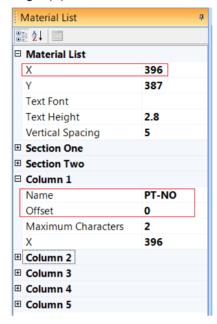
Section Three

Groups layout and format properties for the report data contained in Section 3 of the material list. This category appears on the panel only if **Number of sections** on the **Basic Setup** page is set to **Three**.

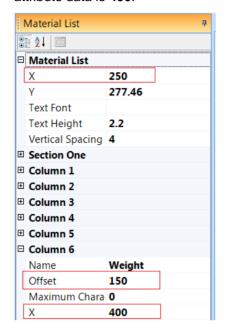
- Content displays the category of data plotted in Section 1 of the material list. Select Fabrication, Erection, Offshore, or Erection/Offshore. The category that you select must not be the same as the categories defined for Section 1 and Section 2. The default setting is Offshore.
- List Down controls the direction in which the material list is printed with reference to the X and Y settings under the Material List category. If you select Down, Isogen plots the material list from top to bottom. If you select Up, the material list is plotted from bottom going up.
 - ★ IMPORTANT For a three section material list, all sections must be plotted in the same direction.
- Maximum Entries sets the maximum number of entries in the material list before an overflow sheet is generated. Type the required value (positive integer).
- X Offset specifies the X (or horizontal) offset for Section 3 of the material list with reference to the report origin defined by the X and Y settings under the Material List category. Type a value (integer) in mm or inches, depending on the Backing Sheet Units setting on the Basic Setup page.
- Y Offset specifies the Y (or vertical) offset for Section 3 of the material list with reference to the report origin defined by the X and Y settings under the Material List category. Type a value (integer) in mm or inches, depending on the Backing Sheet Units setting on the Basic Setup page.

Column Settings

Controls what component attributes are to be plotted in the report, along with basic layout properties. In the example below, the material list is set up to plot five columns of attribute data. The first component attribute (**PT-NO**) is to be plotted with a zero (**0**) offset from the origin (**X**).



- Name specifies the name of the component attribute being defined in the report. Select the required attribute in the list.
- Offset specifies the offset from the X setting (or report origin) defined under the Material List category. Type a value (real number) in mm or inches, depending on the Backing Sheet Units setting on the Basic Setup page. The value that you enter has a direct impact on the column X setting. For example, if the horizontal position of the origin for the first line of report data is set to 250, and you set a 150mm offset from that



origin under **Column 6**, the horizontal start position (or **X** setting) for that component attribute data is **400**.

- Maximum Characters sets the width of the column. The value that you enter must be zero or greater.
- X specifies the horizontal) start position for the column data with reference to the X setting of the material list origin. The value that you enter causes the Offset setting to update automatically. Type a value (integer) in mm or inches, depending on the Backing Sheet Units setting on the Basic Setup page.

■ NOTES

- You can also define Offset and X, Y settings graphically using the mouse. For more information, see Define X, Y, and Offset Settings Graphically in Customize report settings (on page 240).
- Use **Add Column** and **Delete Column** to control which component attributes are plotted in the material list report. For more information, see *Add and Remove Report Columns* in *Customize report settings* (on page 240).

Weld List Page

Defines the layout and appearance of the report data in the weld list that Isogen plots on the isometric drawing. The **Drawing Setup Tool** displays this page only when **User Defined Weld List** is selected on the **Basic Setup** page. The **Weld List** page is divided into two sections.

- Weld List window displays properties that control the report set up and report contents in a
 tree view format. These properties can be customized to meet your specific isometric
 drawing requirements. For more information, see *Customize report settings* (on page 240).
- View window displays a copy of the selected style's default backing sheet with the location of the report data superimposed upon it.

Weld List Window

■ NOTES

- Many of the properties listed below can also be defined or modified on the Basic Setup page of the tool.
- The Weld List window can be displayed, hidden, docked, maximized, tabbed, or made to float. For more information, see Moving and Resizing Windows in Drawing Setup Tool (on page 234).

Categorized

Groups weld list properties in the Weld List window by category.

2↓ Alphabetical

Groups weld list properties in the Weld List window as a flat, alphabetical list.

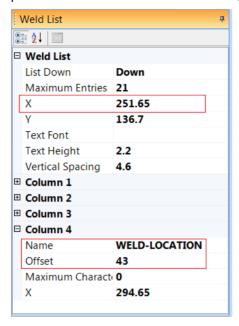
Weld List

Groups layout and format properties for weld list report data, including the report origin and the text size and font.

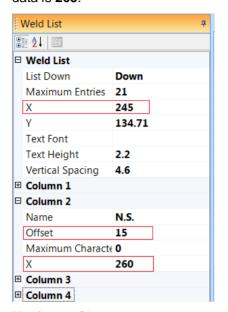
- List Down controls the direction in which the weld list is printed with reference to the X and Y settings under the Weld List category. If you select Down, Isogen plots the weld list from top to bottom. If you select Up, the weld list is plotted from bottom going up.
- Maximum Entries sets the maximum number of entries in the weld list before an overflow sheet is generated. Type the required value (positive integer).
- **X** sets the bottom left X, or horizontal, position of the first line of the weld list that is plotted on the isometric drawing. Type a value (integer).
- Y sets the bottom left Y, or vertical, position of the first line of the weld list that is plotted on the isometric drawing. Type a value (integer).
- Text Font specifies the font used to display weld list report data. Select the required font in the Text Font list.
- Text Height specifies the height of the text that appears in the weld list report. This
 setting applies to all sections of the weld list. Type a value (real number) in mm or
 inches, depending on the Backing Sheet Units setting on the Basic Setup page.
- Vertical Spacing specifies the vertical spacing between the lines of data. The vertical spacing applies to all sections in the weld list. Type a value (real number) in mm or

Column Settings

Controls which component attributes are plotted in the weld list report, along with basic layout properties. In the example below, the plotted weld list is set up to contain four columns of attribute data. The fourth component attribute (**WELD-LOCATION**) is to be plotted with a 43mm offset from the origin (**X**).



- Name specifies the name of the component attribute being defined in the report. Select the required attribute in the list.
- Offset specifies the offset from the X setting (or report origin) defined under the Weld List category. Type a value (real number) in mm or inches, depending on the Backing Sheet Units setting on the Basic Setup page. The value that you enter has a direct impact on the column X setting. For example, if the horizontal position of the origin for the first line of report data is set to 245, and you set a 15mm offset from that origin



under Column 2, the horizontal start position (or X setting) for that component attribute data is 265.

- Maximum Characters sets the number of entries that the column can accommodate.
 The value that you enter must be zero or greater.
- X specifies the horizontal start position for the column data with reference to the X setting of the weld list origin. The value that you enter causes the Offset setting to update automatically. Type a value (integer) in mm or inches, depending on the Backing Sheet Units setting on the Basic Setup page.

■ NOTES

- You can also define Offset and X, Y settings graphically using the mouse. For more
 information, see Define X, Y, and Offset Settings Graphically in Customize report settings (on
 page 240).
- Use **Add Column** and **Delete Column** to control which component attributes are plotted on the weld list report. For more information, see *Add and Remove Report Columns* in *Customize report settings* (on page 240).

Cut List Page

Defines the layout and appearance of the report data in the cut list that is plotted on the isometric drawing. The **Drawing Setup Tool** displays this page only when **User Defined Cut List** is selected on the **Basic Setup** page. The **Cut List** page is divided into two sections.

- Cut List window displays properties that control the report set up, report contents, and report location (shown graphically). These properties can be customized to meet your specific isometric drawing requirements. For more information, see *Customize report* settings (on page 240).
- View window displays a copy of the selected style's default backing sheet with the location of the report data superimposed upon it.

Cut List Window

■ NOTES

- Many of the properties listed below can also be defined or modified on the Basic Setup page of the tool.
- The Cut List window can be displayed, hidden, docked, maximized, tabbed, or made to float. For more information, see Moving and Resizing Windows in Drawing Setup Tool (on page 234).

Categorized

Groups cut list properties in the Cut List window by category.

2↓ Alphabetical

Groups cut list properties in the Cut List window as a flat, alphabetical list.

Cut List

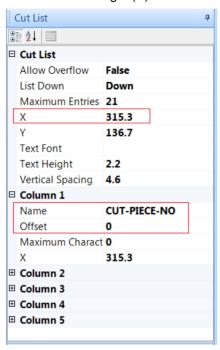
Groups layout and format properties for the cut list report data, including the report origin and the text size and font.

- Allow Overflow controls whether the cut list causes an overflow drawing.
- List Down controls the direction in which the cut list is printed with reference to the X and Y settings under the Cut List category. If you select Down, Isogen plots the cut list from top to bottom. If you select Up, the cut list is plotted from bottom going up.
- Maximum Entries sets the maximum number of entries in the cut list before an overflow sheet is generated. Type the required value (positive integer).
- **X** sets the bottom left X, or horizontal, position of the first line of the cut list that is plotted on the isometric drawing. Type a value (integer).
- Y sets the bottom left Y, or vertical, position of the first line of the cut list that is plotted on the isometric drawing. Type a value (integer).
- Text Font specifies the font used to display cut list report data. Select the required font
 in the Text Font list.
- Text Height specifies the height of the text that appears in the cut list report. This
 setting applies to all sections of the cut list. Type a value (real number) in mm or inches,
 depending on the Backing Sheet Units setting on the Basic Setup page.

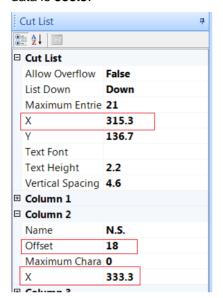
Vertical Spacing specifies the vertical spacing between the lines of data. The vertical spacing applies to all sections in the cut list. Type a value (real number) in mm or inches, depending on the Backing Sheet Units setting on the Basic Setup page.

Column Settings

Controls which component attributes are plotted in the cut list report, along with basic layout properties. In the example below, the plotted cut list is set up to display five columns of attribute data. The first component attribute (**CUT-PIECE-NO**) is to be plotted with a zero offset from the origin (**X**).



- Name specifies the name of the component attribute being defined in the report. Select the required attribute in the list.
- Offset specifies the offset from the X and Y settings defined under the Cut List category. Type a value (real number) in mm or inches, depending on the Backing Sheet Units setting on the Basic Setup page. The value that you enter has a direct impact on the column X setting. For example, if the horizontal position of the origin for the first line of report data is set to 315.3, and you set an 18mm offset from that origin



under **Column 2**, the horizontal start position (or **X** setting) for that component attribute data is **333.3**.

- Maximum Characters sets the number of entries that the column can accommodate.
 The value that you enter must be zero or greater.
- X specifies the horizontal start position for the column data with reference to the X setting of the cut list origin. The value that you enter causes the Offset setting to update automatically. Type a value (integer) in mm or inches, depending on the Backing Sheet Units setting on the Basic Setup page.

■ NOTES

- You can also define Offset and X, Y settings graphically using the mouse. For more information, see Define X, Y, and Offset Settings Graphically in Customize report settings (on page 240).
- Use Add Column and Delete Column to control which component attributes are plotted on the cut list report. For more information, see Add and Remove Report Columns in Customize report settings (on page 240).

Attributes Page

Defines the pipeline attributes that are plotted on the drawing frame. The **Attributes** page is divided into two sections.

- View window displays a copy of the selected style's default backing sheet.
- Attributes window displays the component attributes. Each attribute has a position, and you can define its text size, font, and layer independently. These properties can be customized to meet your specific isometric drawing requirements. For more information, see *Define drawing frame attributes* (on page 245).

Attributes Window

■ NOTE The **Attributes** window can be displayed, hidden, docked, maximized, tabbed, or made to float. For more information, see *Moving and Resizing Windows* in *Drawing Setup Tool* (on page 234).

Attribute

Specifies the name of the attribute being defined.

X

Defines the X, or horizontal, position for the text. Type a value in mm or inches, depending on the **Backing Sheet Units** setting on the **Basic Setup** page.

Υ

Defines the Y, or vertical, position for the text. Type a value in mm or inches, depending on the **Backing Sheet Units** setting on the **Basic Setup** page.

Text Height

Sets the character height for the text. Type the required size in mm or Inches (as a real number), depending on the **Backing Sheet Units** setting on the **Basic Setup** page.

Font

Specifies the font used when plotting the attribute in the drawing frame. Select the required font in the list.

Barcode

Sets the barcode standard used by Isogen. The value you enter creates the start and finish characters for the barcode reader. Type the number that corresponds to the required standard: **1** (Barcode 30), **2** (Barcode 25), **3** (Barcode 25 Interleave), or **4** (Barcode 128). Although several barcode standards exist worldwide to suit different industries, Isogen has only standardized on four.

Justification

Sets the justification for the text. Select **Left**, **Right**, or **Centre**.

Layer

Specifies a number that corresponds to the layer. This number is an integer that identifies the required layer, or level, to which the component type is to be assigned. Select the required value in the list.

Colour

Specify the index number that represents a specific color as defined in the output drawing software. This option is required only for MicroStation or AutoCAD output. AutoCAD uses a fixed 256 color pallet starting at 1 (red) to 256. In MicroStation, in addition to a default color pallet, each design file can have a unique custom color table. For example, color index number 3 may be red in one design file, but it may be green in another. The table below shows a color comparison between a sampling of AutoCAD and MicroStation color index numbers.

AutoCAD Color Index Number	MicroStation Color Index Number	Color Output on Isometric Drawing
1	3	
2	4	
3	2	
4	7	
5	1	
6	5	
7	0	
8	9	
9	14	

NOTE For specific information about AutoCAD, MicroStation, and other 3rd party drawing software color tables, refer to the product documentation delivered with your software.

Char Width

Sets the character width for the text. Type the required size in mm.

Weight

Controls the character thickness. Type a value between 1 and 9. This option is only used for MicroStation.

Rotation

Defines the angle of rotation in clockwise degrees.

Truncation Length

Defines the number of output characters.

NOTE Use **Add / Remove Attributes** [†] on the vertical toolbar to control which drawing frame attributes are plotted on the isometric drawing. For more information, see *Define drawing frame attributes* (on page 245).

Add/Remove Attributes Dialog Box

Lists the attributes that are plotted in the drawing frame, as well as the attributes that are available to be plotted. The software opens this dialog box when you click **Add / Remove**

Attributes †- on the Drawing Setup Tool vertical toolbar.

Available

Lists the attributes that you can select to plot on the isometric drawing.

In Use

Lists the attributes that are currently selected for plotting on the isometric drawing.

→ Add

Moves the selected attribute to the In Use list.



Remove

Removes the selected attribute from the In Use list.

OK

Closes the dialog box, and returns you to the **Attributes** page. The attributes that you selected are displayed in the **Attributes** window grid area.

Cancel

Closes the dialog box without making any changes to the grid in the Attributes window.

NOTE The **Available** and **In Use** lists are standard multi-select lists, allowing you to select one or more attributes. To select consecutive attributes, select the first attribute, press and hold down the SHIFT key, and then select the last attribute. To select nonconsecutive attributes, press and hold down the CTRL key, and then select each attribute.

SECTION 20

Alternative Text

Alternative Text, or AText as it is often called, is a powerful Isogen feature that enables you to easily change or remove any text on the isometric drawing. The feature operates by assigning a unique identification number to each standard text string. The system then refers to this ID number whenever a change is required to the text string that the number represents.

By definition, a standard text string can be a single character, a single word, or a group of words. However, some ATexts are, by default, set to an all-blank word. The total number of separate standard text strings held by the software in this way is in excess of 300.

Identification Number Format

Because identification numbers are always negative, the numbers must be preceded by a minus sign. For example, AText -249 represents the default word **WEST**, which is used in the main isometric drawing area.

Using Alternative Text

Using Alternative Text

AText allows you to substitute your own text terminology or language in place of the standard Isogen words on the isometric. To make a word change, you do not have to replace the entire standard AText. You can change as little as a single one word, if that is all that is required.

Although the AText feature has a considerable degree of built-in flexibility, you must exercise a certain amount of care when defining your own words, particularly in terms of word lengths. As a general rule, newly defined words or word strings should be about equal in length or shorter than the text you are replacing. Obvious exceptions to this are the cases of the single line headings in the material list region. In this respect, you must take full responsibility for word definition. Isogen does not warn you in cases which words are too long and cannot be accommodated in the standard space provided on the isometric. Badly designed AText can lead to undesirable results, such as over-writing or incorrectly positioned text.

Special Characters

The following special characters further extend the functionality of ATexts:

- The Dollar (\$) Sign The \$ character, which is used in Isogen to force a new line in regular isometric message text, can also be used with ATexts. However, when using the \$ character, we recommend that you carefully check the output results of each occurrence.
- The Question Mark (?) The ? character has the following two uses:
 - In ATexts -210 (Flange Part Number), -211 (Gasket Part Number) and -212 (Bolt Part Number) to suppress the plotting of the single characters normally associated with these ATexts, without switching off their associated facility, as would normally happen when an AText is set to blank. For example, setting AText -210 ? suppresses the F character that is usually plotted. This AText does not completely suppress the plotting of the flange material list part number as would normally happen when an AText item is set to blank. Using the ? character is particularly useful on spool isometrics.
 - In some special ATexts at points where the software dynamically inserts information. By default, AText -456 is set to DETAIL?. As a result, the software inserts a letter or a number, depending upon the system that you have specified at the position of the? character.
- The "At" (@) Sign You can use the @ character to pad out an AText string with trailing blanks. As a result, the text that follows the AText is separated from the AText by a series of blanks.

Foreign Language Use

AText is particularly beneficial to foreign language users who want to produce isometrics containing text in their native language. However, there are restrictions governing which characters are permissible. Those restrictions are outlined in *The Usable Character Set*.

The Usable Character Set

You can only use the following Standard English and special purpose characters in AText definitions:

- Upper- or lower-case letters in the range A to Z.
- Numeric characters in the range 0 to 9 inclusive.
- A blank space character.
- The following special purpose symbols: * + . , : [] () # ' < > = | & %
- ★ IMPORTANT Other foreign language characters, such as those used in the Cyrillic, Greek, or Chinese alphabets, are excluded, as are specially accentuated characters, such as à, á, â, and so on.

AText and the Drawing Frame Symbols

The AText feature goes further than just controlling text characters. The standard symbols appearing in the line summary area across the bottom of the standard Isogen drawing frame, for shop weld, field weld, and so on through traced pipe, can all be suppressed when they are not required by setting their associated ATexts to blank.

Composite Text Messages

Composite text messages are made up of more than one text item. The composition is done by Isogen automatically. Such messages can be composed in either of the following two ways:

- By combining two or more related ATexts
- By combining AText and an associated design database attribute value

Generally, in cases in which AText operates together with design database attribute information to form a composite message, setting the AText part to blank to suppress plotting the message causes suppression of the attribute text as well. For example, if the composite message **BATCH REF: 12/100A/C** in the title block area needs to be completely suppressed, then setting AText - 252 (**Title Block Batch**), which contains the words **BATCH REF**, to blank causes both this and the attribute part, **12/100A/C**, not to be plotted.

See Also

Examples (on page 270)

Examples

The following are some examples of standard ATexts:

- The material list heading ERECTION MATERIALS.
- The isometric connection messages CONN. TO and CONT. ON in the main drawing area.
- The BATCH REF and PIPING SPEC headings in the title block area.

Each of these are default AText words that are programmed in by Isogen. However, you can reprogram them, if required, as explained in the following examples.

Example 1: ERECTION MATERIALS

You can find ERECTION MATERIALS as AText -310 (Erection Material Handling). To change the default heading (ERECTION MATERIALS) to CONSTRUCTION MATERIALS, the entry -310 CONSTRUCTION MATERIALS must appear in the appropriate data input file.

Example 2: CONN.TO

CONN TO. is an example of a composite message that is used at locations at which pipelines are connected to equipment nozzles. You can find it as AText -208 (Equipment Connection).

To change the message CONN. TO to JOIN TO, the entry -208 JOIN TO must appear in the appropriate data input file. Information regarding the joined to component, that is, the nozzle name as extracted from the design database, is automatically appended to the AText by Isogen to form a composite message, such as **JOIN TO D45-NZ12**.

Example 3: BATCH REF

BATCH REF is another composite message example (AText -252 (Title Block Batch). The message is used to convey plant zone or area information in the title block area of the isometric where the contents of the BATCH (AREA) type record in the pipeline input data file is automatically appended by Isogen. An example of this kind of message is BATCH REF: ARATE/N12.

To remove the batch reference entry from the isometric, set the AText entry in the input data file (IDF) to blank, thus: **-252**.

Doing so removes the entire composite message from the isometric, both the AText BATCH REF : part and the following design database attribute that holds the batch reference information.

NOTE You can quickly locate a specific AText using the Search feature. For more information, see *Search Filters* (on page 160).

SECTION 21

Search Folders

Search Folder allows you to search for documents based on common properties such as out-of-date status, approval, or documents that have been published to a certain contract in

integrated environment. You create a search folder by right-clicking the root model or a folder in the **Management Console** or the **Drawing Console** and selecting **New > Search Folder**.

After running the query defined for a search folder, you can perform such tasks as **Update** or **Publish** as if you are working from the actual component for the documents.

The documents found by a search folder query are listed in the **Detail View**. You are able to interact with the search folder documents just as if you were dealing with the actual components that own the documents. Search folders can also be used for reports.

Search Folder Shortcut Menu

Right-click a search folder component uto display the shortcut menu.

Run Query

Runs the query specified by the search folder setup definition. If you have not run the search folder **Setup** command, this command is not available.

Publish

Publishes a single document or all documents in the imported folder. This command is only available if the model is registered with SmartPlant Foundation.

Update and Publish

Updates and immediately publishes a single document or all documents in the search folder. This command is only available if the model is registered with SmartPlant Foundation.

Setup

Specifies the query to run for the search folder. For more information, see *Setup (Search Folder)* (on page 274).

NOTE Setup for the search folder does not perform the individual setups for any of the drawing by query documents found in the search folder query and shown in the detail view.

Copy

Copies the search folder. It does not copy the associated documents shown in the detail view area. The search folder setup information is saved with the copy. After you paste the copy to a different location in the **Console** hierarchy, you can run **Setup** again as needed for the new search folder.

Delete

Deletes the search folder. It does not delete any of the documents found in the associated detail view.

Rename

Renames the search folder. It does not affect any of the documents found in the associated detail view.

Revise

Allows you to revise all documents in the search folder without publishing them. This command is only available if the model is registered with SmartPlant Foundation. For more information, see *Revise* (on page 513).

Save Package

Allows you to save the search folder and its definition as a package to be reused in other folder locations in the hierarchy. If you have not run the search folder **Setup** command, this command is not available.

Properties

Displays the Configuration Properties for the document.

NOTE To change properties on the document, go to the root location of the document. The search folder does not participate in any propagation of properties to its documents.

■ NOTES

- After a search folder is placed in the console, it follows the same localized naming convention as a folder, such as "New Search Folder" and "New Search Folder (2)."
- Unlike a folder , you cannot create child components in a search folder . This includes pasting existing components; however, you can copy the search folder and paste it elsewhere in the hierarchy.
- Performing Publish, Update and Publish, and Revise on a folder does not include documents under a child search folder. The commands need to be performed separately on the child search folder in order for its documents to be affected.
- You can run commands from the search folder level to modify all of the documents within the folder, or you can modify the individual documents by selecting them in the **Detail View**.

Create a search folder

1. Right-click the model root or a folder in the **Management Console**, and select **New** > **Search Folder**.

The search folder is added to the hierarchy.

2. Right-click on the new search folder, and select Setup.

The **Setup** dialog box displays.

- 3. Select **More** in the **Filter** list, and select a filter. Click **Properties** of to display the **Filter Properties** dialog box.
- 4. Select **More** in the **Start From** list, and select a component from the **Select Drawings Component** dialog box.
- 5. Click OK.
- 6. Right-click the search folder, and select Run Query.

The documents returned by the query are added to the Search Folder listing in the **Detail View**.

7. To rename the folder, right-click the folder, and select **Rename**, or select the folder, and press **F2** on the keyboard. Type a new name.

Setup (Search Folder)

Sets options for creating a search folder. This command is available on the **Search Folder** shortcut menu and displays the **Setup** dialog box.

NOTE Setup for the search folder does not perform the individual setups for any of the drawing by query documents found in the search folder query and shown in the detail view.

Setup Dialog Box (Search Folder) (on page 275)

Setup Dialog Box (Search Folder)

Specifies a filter that identifies the objects to be included in the search folder query. The **Setup** definition tells the query "where" to look for the objects specified by the component "what" filter.

Filter

Identifies the filter that defines the *what* portion of the query. The software uses the filter to determine the objects included in the drawings when they are generated. Select **More** in the **Filter** list to display the **Select Filter** dialog box. Click **Properties** of to display the **Filter Properties** dialog box. For more information, see *Search Folder Filters* (on page 275).

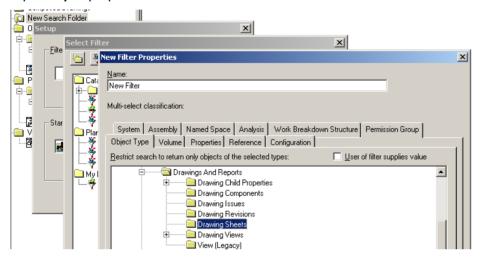
Start From

Specifies the location in the model from which to start the object search. This is the "where" side of the query. Select **More** in the **Start From** combo box to display the **Select Location** dialog box.

Search Folder Filters

Search folders use filters to specify how the component identifies the documents to include in the search folder. You can use Drawings and Reports object properties to define filters when creating search folder components. This allows you to search for documents based on common properties such as out-of-date status, approval, or documents that have been published to a certain contract in integrated environment.

When you run **Setup** on a search folder, you can create filters that check for specific drawing or report object properties.



The following examples show how you might create filters to search for specific drawing object properties:

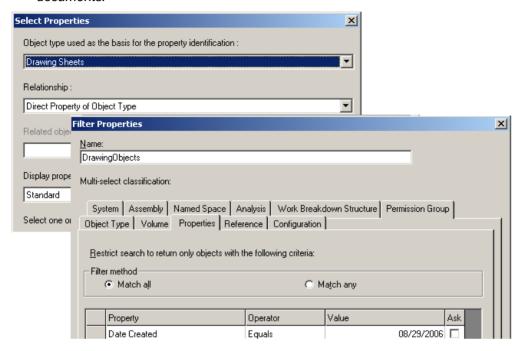
Basic Drawing Document Properties - Title, Area, and Signature

You can use the **Drawing Sheet** object and its properties to look for properties associated with the drawing documents. The following procedure shows how to access the Title, Area, and Signature properties on the drawing and report documents.

- On the Properties tab of the Filter Properties dialog box, select More in the Property field.
 The Select Properties dialog box displays.
- Under Object type used as the basis for the property identification, select Drawing Sheets.

3. Under Relationship, select Direct Properties of Object Type.

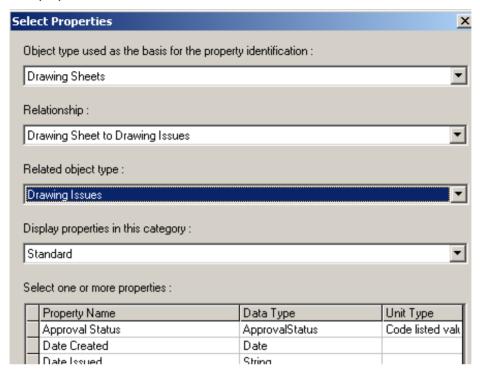
You can now search the documents for specified properties under **Select one or more properties**. For example, you can search specifically for the **Date Created** value on the documents.



Issue or Revision Properties

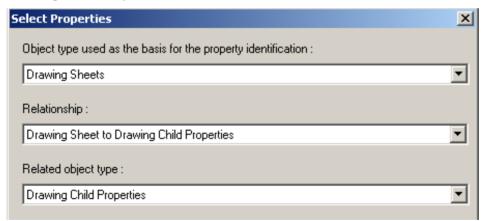
To search specifically for issue or revision properties on the drawing sheet or drawing component, set the **Relationship** to **Drawing Sheet (or Drawing Component) to Drawing**

Issues or **Drawing Sheet (or Drawing Component) to Drawing Revisions** when setting the filter properties.



Baseline, Style, Smart 3D, or Custom Attribute Properties

To search specifically for baseline, style, Smart 3D, or custom attribute properties, set the **Relationship** to **Drawing Sheet to Drawing Child Properties**, and set **Related object type** to **Drawing Child Properties**.



See Also

Setup Dialog Box (Search Folder) (on page 275)

SECTION 22

Imported Folders

Imported Folder allows you to import an external Windows folder containing any type of file available in Windows. You create an imported folder by right-clicking a folder in the **Management Console** or the **Drawing Console** and selecting **New** > **Imported Folder**.

Imported files are opened using their standard Windows default software. For example, a .docx file opens in Microsoft Word.

Imported Folder Shortcut Menu

Right-click an imported folder component to display the shortcut menu.

Delete

Deletes the imported folder and its files from the database. Deleting does not affect the original folder and files on the network drive.

Rename

Renames the imported folder. Renaming does not affect the files in the folder or the name of the original folder on the network drive.

Revise

Allows you to revise all documents in the imported folder without publishing them. This command is only available if the model is registered with SmartPlant Foundation and when the documents are persisted in the model database. For more information, see *Revise* (on page 513) and *Setup Dialog Box (Imported Folder)* (on page 282).

Export

Exports files in the imported folder to the specified folder path. This command is only available after setup of the imported folder, and when the folder and files are persisted in the database.

Refresh

Refreshes the status of files in the imported folder when the date of a file is earlier than the date of the file the on the network drive. The icon of an out-of-date file is superimposed with

This command is only available after setup of the imported folder.

Publish

Publishes a single document or all documents in the search folder. This command is only available if the model is registered with SmartPlant Foundation and when the documents are persisted in the model database. For more information, see *Setup Dialog Box (Imported Folder)* (on page 282).

Update and Publish

Updates and immediately publishes a single document or all documents in the search folder. This command is only available if the model is registered with SmartPlant Foundation and when the documents are persisted in the model database. For more information, see *Setup Dialog Box (Imported Folder)* (on page 282).

Setup

Specifies the folder to import. For more information, see *Setup (Imported Folder)* (on page 281).

Update Now

Updates all files in the imported folder to match the files on the network drive. This command is only available after setup of the imported folder.

Properties

Displays the properties of the folder.

Imported Files

Right-click an imported file to display the shortcut menu.

Open

Opens the file with the appropriate Windows application.

Properties

Displays the properties of the file.

Update

Updates the file to match the files on the network drive. This command is only available after setup of the imported folder.

★ IMPORTANT Select a shared network folder (with a path beginning with \\) instead of a local folder to allow refresh and update of file changes between users.

NOTE Unlike a folder , you cannot create or paste other components within the imported folder .

Create an imported folder

1. Right click a folder in the **Management Console**, and select **New > Imported Folder**.

The imported folder is added to the hierarchy.

2. Right-click the folder, and select Setup.

The **Setup** dialog box displays.

3. Type the path in the **Folder to Import** box.

OR

Click **Browse** , navigate to the needed folder in the **Browse for Folder** dialog box, and click **OK**.

- 4. In the **Setup** dialog box, click **Persist in Database** to add the imported files to the model database.
- 5. Click OK.

The imported folder and its files are added to the hierarchy.

6. To rename the folder, right-click the folder, and select **Rename**, or select the folder, and press **F2** on the keyboard. Type a new name.

Setup (Imported Folder)

Sets options for importing an external folder and its files. This command is available on the **Imported Folder** shortcut menu and displays the **Setup** dialog box.

Setup Dialog Box (Imported Folder) (on page 282)

Setup Dialog Box (Imported Folder)

Specifies a folder containing the files to import.

You should import from a shared network folder having a path beginning with \\ to allow everyone to have access to the same files and to avoid multiple file copies or unintended overwriting of changes from another user. Windows handles read-only locking of the files when they are open. You cannot import the %Temp% folder because it is used for file processing during import.

Folder to Import

Specifies the folder path. Type a path, or click **Browse** in to select a path.



Allows you to browse for a folder location. For more information, see *Browse for Folder Dialog Box* (on page 282).

Persist in Database

When selected, adds imported files to the model database. When cleared, only the folder path to the files is imported.

NOTE Export is not available when Persist in Database is cleared.

Browse for Folder Dialog Box

Displays a hierarchical list of folders. Select a folder from this list, and click **OK** to choose it, or double-click a folder to display any sub-folders. The selection you make on this dialog box appears in the **Folder to Import** box of the **Setup** dialog box.

Make New Folder

Creates a new folder in the hierarchy, beneath the currently selected folder.

SECTION 23

Tools Menu

Several tools are provided within the Drawings and Reports task. These tools display on the **Tools** menu in the Drawings and Reports task.

Reference Data Tools

The following commands on the **Tools** menu are for use by your reference data administrator. They apply strictly to customizing orthographic drawing and marine mode drawings by rule reference data:

- Define View Style
- Define Layout Style
- Edit Border Template
- Edit Layout Template

You do not use these commands in the normal workflow of creating drawings and reports. For more information, see the *Drawings and Reports Reference Data Guide*.

Other Tools

You can use the **Custom Command** tool to set up special macro commands you use in your documents. **Batch Management** processes drawing updates and printing on a batch server. **Convert Legacy Snapshots** converts legacy version 6.1 snapshot drawings to composed drawings. For more information, see the *Orthographic Drawings User's Guide*.

In This Section

Batch Processing - Intergraph Smart Batch Services	283
Batch Processing	
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Batch Processing - Intergraph Smart Batch Services

With batch processing, you can update, print, or refresh your documents without dedicating your computer to the operation.

Drawings Batch Dialog Box (on page 284)

What do you want to do?

- Add Actions to Queue (on page 285)
- Create a Template (on page 286)

Drawings Batch Dialog Box

Updates, prints, and refreshes document batch jobs without requiring a dedicated computer for the operation.

Available Actions

Lists the actions that you can perform within a batch job. The actions available are based on permissions and document type.

- Refresh Refreshes documents on a batch queue.
- Full Update Updates all drawings by rule documents, regardless of the current status
 of the documents. This action is only available in the Marine mode.
- Update Updates documents on a batch gueue.
- Save As Specifies the file format to which to save Smart 3D drawings.
- Print Schedules a document to be printed from a batch queue.

Actions to Queue

Lists the actions that are performed in the batch job.

- Remove Removes an action from the queue.
- Clear Removes all actions from the gueue.

Action Options

Defines the parameters of an action. The displayed options depend on the selected action.

Template Name

Saves print settings as a template. Type a name in the box, and click **Save Settings as Template.** While optional, creating a new template is a quick method to submit a batch job with the specified options.

Save Settings as Template

Creates a user-parameterized action.

Delete

Removes a template from **Available Actions**. This option is available only on user-created templates.

Save As

For more information on the **Save As** action options, see *Save As Dialog Box* (on page 78).

NOTE In the **Output Folder** field, select a shared folder through its UNC path. A UNC path has the general form of \\server\share.

Print

- **Printer** Displays all printers configured on the client computer. The name of the printer on the batch queue must match for the print to be successful.
- Copies Specifies the number of copies to print.
- Black and White Prints a black and white drawing, if checked.
- Orientation Indicates the orientation of the printed output. Select Portrait or Landscape.
- Paper Size Displays the paper sizes supported by the selected printer.

Use 64-bit if available

Updates drawings with 64-bit processes. This option is only available for **Update** actions performed on composed drawings.

■ NOTES

- If you select **Use 64-bit if available** and run **Update** on a folder component containing composed drawings and other drawing types, the composed drawings are updated with 64-bit processes. All other drawings are updated with 32-bit processes.
- You cannot update a composed drawing with 64-bit if the drawing contains a view style that includes PDS reference data.

Schedule

Creates the job and opens the **Schedule Batch** dialog box.

NOTE If you click **Schedule** with no actions in **Actions to Queue**, you receive a message, and the dialog box remains open.

Add Actions to Queue

You can set up batch jobs for a single document or for multiple documents.

Single document:

- Right-click on a single drawing either in the **Detail View** or in the **Drawings Console**, and select **Batch**.
- 2. Select the action, and click **Add** to move it into **Actions to Queue**.

■ NOTES

- You can add actions to the queue in any order, but the software processes the actions in the following order: Refresh, Update, and then all other actions. If you add Update, but you do not add Refresh, Update is the first action processed.
- You can only add one Save As action to the queue.
- 3. Define any action options. For more information, see *Batch Processing Intergraph Smart Batch Services* (on page 283).
- 4. Click **Schedule** to create the job and open the **Schedule Batch** dialog box.

Multiple documents:

- 1. Do one of the following:
 - Right-click a set of multi-selected documents or components in the **Detail View**, and select **Batch**.
 - b. Right-click a component in the **Management Console**, **Detail View**, or **Drawings Console**, and select **Batch**.
- 2. Select the action, and click **Add** to move it into **Actions to Queue**.

■ NOTES

- You can add actions to the queue in any order, but the software processes the actions in the following order: Refresh, Update, and then all other actions. If you add Update, but you do not add Refresh, Update is the first action processed.
- You can only add one Save As action to the queue.
- 3. Define any action options. For more information, see *Batch Processing Intergraph Smart Batch Services* (on page 283).
- 4. Click **Schedule** to create the job and open the **Schedule Batch** dialog box.

Create a Template

Templates are user-parameterized actions that are saved in the session file and available for future batch jobs. You can only create templates based on the **Print** action.

Create a new template:

- 1. Select the **Print** action, and click **Add** to move it to **Actions to Queue**.
- 2. In Action Options, make the necessary changes to the action.
- 3. Type a new **Template Name** for the action.
- 4. Click Save Settings As Template.

Delete a template:

- 1. Select the template to delete.
- 2. In Action Options, click Delete.

Schedule [Task] Dialog Box

Queue

Displays the name of the queues configured by an administrator for the job. For more information on configuring the queues, see *Configure Queues for Jobs* in the *Project Management User's Guide*.

Run job

Sets the frequency with which the job runs. Jobs can be scheduled to run once or on a regular interval (daily, weekly, or monthly). Depending on the job frequency selected, additional controls display. These controls allow you to define more specific scheduling information. The scheduling controls can be changed only at job submission.

Run on

Sets the time to start running the job.

Options

Opens the *Optional Schedule Properties Dialog Box* (on page 289) that you can use to define a start and end date.

Run on box

Contains a calendar from which you can select the run date. This option is available when you select **Once** from **Run job**.

Every X days

Specifies how many days pass between job runs. This option is available when you select **Daily** from **Run job**.

Every X weeks

Specifies how many weeks pass between job runs. In addition, you can select on which days the job runs. This option is available when you select **Weekly** from **Run job**.

Day X of the month

Specifies on which day of the month the job runs. This option is available when you select **Monthly** from **Run job**.

The X Y of the month

Specifies on which day of the month the job runs. For example, you can select the last Monday of the month. This option is available when you select **Monthly** from **Run job**.

Job Start

Notifies you when the job starts, if Outlook is set up.

Job Completion

Notifies when the job completes, if Outlook is set up.

Job Abort

Notifies you if the job aborts, if Outlook is set up.

Address Book

Selects the name of the person to be notified by e-mail of the job status, if Outlook is set up. If Outlook is not available, this option does not work. You can also type the address manually. The person you define here receives an email with the job log files after the job finishes.

■ NOTES

- The Batch Services SMTP option must be configured on the batch server for this to work. For more information, see the Intergraph Smart Batch Services documentation.
- The WinZip application is no longer required on the batch server to compress any emailed attachments. Compression is now done with functionality included in Smart 3D.

Optional Schedule Properties Dialog Box

Provides more options on the **Schedule Backup** dialog box. This dialog box opens when you click **Options**.

Start date

Sets an optional start date.

End date

Sets an optional end date, if checked.

Batch Processing

With batch processing, you can make sure your documents are updated, printed, or refreshed without having to dedicate your workstation to the operation.

Batch Updating

Using **Batch > Update**, available on the **Drawings Batch** dialog box, you instruct the software to update documents on a Batch Server while you continue to work on other tasks.

■ NOTE Before using **Batch > Update** for 3D Model Data documents, you must set the appropriate surface styles and aspects for the model data. For more information, see *Set surface styles and aspects for 3D model data documents* (on page 47).

Batch Printing

Using the **Batch > Print** command available on the **Drawings Batch** dialog box, you can schedule batch printing jobs as needed to free up valuable processing time.

Batch Refreshing

Using the **Batch > Refresh** command available on the **Drawings Batch** dialog box, you can schedule batch refresh jobs for multiple drawing documents.

Managing Batch Jobs

Tools > **Batch Management** allows you to view your batch jobs and make changes. For more information, see *Manage batch jobs* (on page 292).

Setting Up Batch Processing

Batch Server - The computer on which the batch process runs is called the Drawing Batch Server. The server must have Windows 2000 or Windows 7 and Smart 3D Workstation loaded. The computer designated as the batch server is usually one that is not being used to perform daily tasks, as the process of updating large numbers of documents and drawings can consume a great deal of the computer's resources. You can have one or more batch servers per site database. For more information, see *Project Size Estimates* and *Drawing Batch Server* in the *Intergraph Smart* 3D *Installation Guide*.

★ IMPORTANT To initially configure the Batch Server, you must be an administrator on that computer and have write permissions or better on the model, the SharedContent share, and any permission groups that access drawings.

Client - The workstations that send batch processes to the server are called clients.

■ NOTES

- If the Batch commands are not available on the shortcut menus for your documents, you are not configured to use batch processing
- Most of the scheduling is stored on the Batch Server in the form of scheduled items in the Windows Task Scheduler. After models have been assigned to the Batch Server, new processes display in the Processes tab of the Task Manager dialog box on that computer. The Batch Manager process indicates that at least one model can use this computer as a Batch Server. For each model selected on the Setup 3D Drawings Batch Server dialog box, one Batch Server process displays in the list. If the Batch Manager or Batch Server processes are stopped, the computer does not process batch updates.
- You can also have a Batch Tier process running for each of the selected models. This
 process is created when the Batch Server process finds a batch job and terminates
 automatically after the Batch Server has been inactive for a while.

See Also

Updating Documents (on page 82) Batch Update (on page 299) Batch Print (on page 298) Batch Refresh (on page 301)

Batch Management

Allows you to view your batch jobs and make changes.

What do you want to do?

- Manage batch jobs (on page 292)
- Edit or delete batch jobs (on page 292)
- Configure batch processing in the Common User's Guide
- Remove a model from the batch server in the Common User's Guide

Manage batch jobs

You can only modify or delete batch jobs that you own. You cannot change or delete batch jobs owned by others.

- Select Tools > Batch Management. The Batch Management dialog box displays, showing the currently scheduled batch jobs with their status. For more information, see Batch Management Dialog Box (Batch Management Command) (on page 294).
- 2. Click a column header to sort the batch job table by the column definition.
- 3. Use the View menu items to Refresh the batch job list or toggle the Status Bar on or off.
- 4. Select one or more batch jobs, and then select **Action > Pause** to suspend the idle jobs. This command has no effect on jobs that have started processing.
- 5. Select one or more batch jobs, and then select **Action > Resume** to continue processing the paused jobs. This command has no effect on jobs that are already processing.
- 6. To cancel a batch job, select it in the table, and then select **Action > Cancel**.
- To modify the batch schedule definition for a batch job, select it, and then select Action >
 Properties. The Properties dialog box displays, showing the current schedule definition.
 For more information on modifying the properties, see Properties Dialog Box (Batch Management Command) (on page 296).

TIP You can also right-click a batch job to display a shortcut menu for the **Pause**, **Resume**, **Cancel**, and **Properties**.

Edit or delete batch jobs

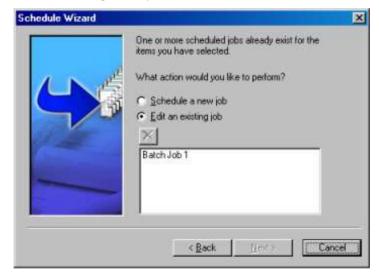
Complete the following steps to edit or delete an existing batch job.

NOTE You can also use **Tools > Batch Management** to update existing batch jobs. For more information, see *Manage batch jobs* (on page 292).

1. Right-click a document that has a batch job scheduled and select the batch command to run: **Update**, **Print**, or **Refresh**. The **Schedule Wizard** displays.



2. The default setting is **Schedule the Batch Job**. Click **Next** to schedule a new batch job or edit the existing batch job.



- 3. To edit or delete an existing batch job, select the **Edit an existing job** option. The table at the bottom of the wizard page enables.
 - **NOTE** To schedule a new batch job for this document, select the **Schedule new job** option and click **Next** to display the next page of the **Schedule Wizard** and create a new batch job schedule. For more information, see *Set Batch Job Frequency (Schedule Wizard)* (on page 305).
- 4. Select a batch job in the table. To delete the batch job, click **Delete** ★. To edit the batch job, click **Next** to display the next page of the **Schedule Wizard** and edit the batch job properties. For more information, see *Set Batch Job Frequency (Schedule Wizard)* (on page 305).

Batch Management Dialog Box (Batch Management Command)

Displays a list of jobs that have been submitted to the batch server, and allows you to view or manage those jobs. You can open this dialog box by selecting **Tools > Batch Management**.

TIP Click a column header to sort the batch job table by the column definition.

Job Name

Displays the name of the batch job.

NOTE Non-scheduled batch jobs use the name of the drawing or drawing component type being processed.

Server

Displays the name of the batch server processing the batch job.

NOTE Batch jobs are submitted to a queue on the model database. The batch servers retrieve the jobs from the queue in a first-in/first-out order.

Parent Component

Identifies the name of the parent component for the batch job.

Request

Displays the type of the batch job request. For example, **Update** or **Print**.

Status

Indicates the current status of the batch job. For example: **Updating**, **Submitted**, **Scheduled**, or **Printing**.

Owner

Displays the name of the owner of the batch job.

Submitted

Shows the date and time that the batch job was submitted or scheduled.

Scheduled

Indicates how the batch job has been scheduled. For example: **Daily**, **Weekly**, **Monthly**, **One time only**, and so forth.

Action Menu

Pause

Suspends the selected idle jobs. A job that is updating will not pause.

Resume

Removes the hold on the paused items. This command has no effect on jobs that are already processing.

Cancel

Deletes the selected jobs from the queue.

Properties

Displays the **Properties** dialog box for the selected job. This command is inactive if multiple jobs are selected. For more information, see *Properties Dialog Box (Batch Management Command)* (on page 296).

View Menu

Status

Turns the display of the status bar on/off.

Refresh

Refreshes the display of the batch job list.

See Also

Batch Management (on page 291)

Properties Dialog Box (Batch Management Command)

Displays the current batch schedule for the selected batch job, and allows you to modify the schedule. You can open this dialog box by selecting **Action** > **Properties** in the **Batch Management** dialog box.

The information on this dialog box changes depending on the active schedule definition of the batch job. The active schedule definition is shown at the top of the dialog box.

Schedule task

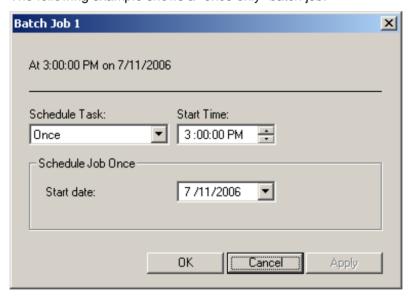
Sets the type of schedule for the task. If you change this setting, the other options on the dialog box change as well.

Start time

Specifies the time for the batch job to start. You can select a time using the scroll button or define a time in the format shown.

Batch Jobs Scheduled Once

The following example shows a "once only" batch job:

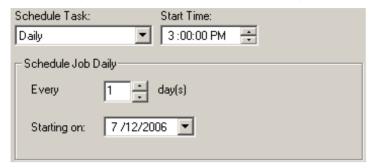


Start date

Specifies the date on which the batch job update begins.

Batch Jobs Scheduled Daily

If the batch job is scheduled to run daily, the Properties dialog box displays as follows:



Every count day(s)

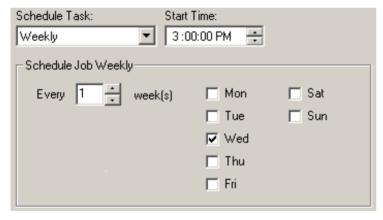
Specifies a number of days. The batch job runs once per the specified count of days. For example, if you specify **2** as the value, the batch job runs once every two days.

Starting on

Specifies the date on which the batch job schedule begins.

Batch Jobs Scheduled Weekly

If the batch job is scheduled to run weekly, the **Properties** dialog box displays as follows:



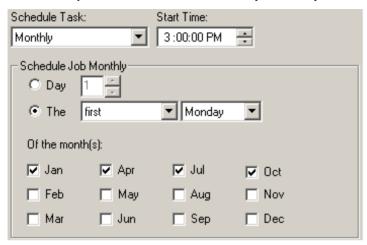
Every count week(s)

Specifies a number of weeks for which the batch job will run.

You can check as many days during the week as required. The batch job runs once per day for the specified count of weeks.

Batch Jobs Scheduled Monthly

If the batch job is scheduled to run monthly, the **Properties** dialog box displays as follows:



You use the two options to specify when the batch job runs during the month. You can check as many months a required. The batch job runs once on the specified day of the specified month(s).

See Also

Batch Management (on page 291)

Batch Print

Allows you to submit document files directly to a print queue for a printer or schedule the document files to be sent to the print queue at a later date or at recurring intervals. You can set up batch print jobs for a single document or a select set of documents you select in the **Detail View**. This command is available when you right-click on a single drawing or a select set of drawings in the **Detail View**.

The **Batch > Print** command displays the **Schedule Wizard** so you can specify whether the drawing should be printed now or at a scheduled date and time. The batch print job runs on a batch server. If the command is not available on the shortcut menu for your documents, batch processing is not configured.

To walk-through the **Schedule Wizard** and set up batch printing, see *Batch Print Schedule Wizard Common Tasks* (on page 310).

To view and manage current batch jobs, see *Manage batch jobs* (on page 292).

■ NOTES

- After setting up the batch print job, the software checks to see if any of the documents are being updated. If so, the software holds the print job until the updates are complete.
- As long as a document has a file created for it, you will be able to print the drawing. The software does not check to see if the drawing is up-to-date, out of date, or in an error state.
- The Batch Print command is not available for a MicroStation 3D DGN component document file.
- The drawing is printed on the default printer of the batch server that processes the print job.

Batch Update

Allows you to update documents on a Batch Server while you continue to work on other tasks. You can set up batch update jobs for a single document or a select set of documents you select in the **Detail View**. This command is available when you right-click on a single drawing or a select set of drawings in the **Detail View**.

The **Batch > Update** command displays the **Schedule Wizard** so you can specify whether the drawing should be updated now or at a scheduled date and time. The batch update job runs on a batch server. If the command is not available on the shortcut menu for your documents, batch processing is not configured.

To walk-through the **Schedule Wizard** and set up batch updates, see *Batch Schedule Wizard Common Tasks* (on page 302).

To view and manage current batch jobs, see Manage batch jobs (on page 292).

■ NOTES

- Before using Batch > Update for 3D Model Data documents, you must set the appropriate surface styles and aspects for the model data. For more information, see Set Surface styles and aspects for 3D model data documents (on page 47).
- The Batch > Update command detects when only border changes have been made and only updates the border portion of the drawing that is out-of-date.
- If the software cannot make a SmartPlant Foundation server connection when you use Batch > Update for 3D Model Data documents, you are prompted to provide a valid login and password.

See Also

Updating Documents (on page 82) Batch Processing (on page 289)

Batch Local Update

Allows you to update documents locally while you continue to work on other tasks. This command is available when you right-click on a single drawing. The command triggers a local background update against the drawing document.

To view and manage current batch jobs, see *Manage batch jobs* (on page 292).

■ NOTES

- Before using Batch > Local Update for 3D Model Data documents, you must set the
 appropriate surface styles and aspects for the model data. For more information, see Set
 surface styles and aspects for 3D model data documents (on page 47).
- The default timeout value for updating documents through the Batch Server is 40 minutes. For more information on setting the Batch timeout (in minutes) property on a drawing document, see Style Tab (Properties Dialog Box) (on page 55).
- You cannot process more than one Batch Local Update at a time. If you attempt to process a second Batch Local Update, an error dialog will display. For more information, see Updating Documents (on page 82).
- If the software cannot make a SmartPlant Foundation server connection when you use Batch > Update for 3D Model Data documents, you are prompted to provide a valid login and password.

Batch Update Document(s)

Batch > **Update Document(s)** updates existing drawings or reports if they are out-of-date. This command updates all the documents associated with a drawing or report component. You can also multi-select documents within the **Detail View**.

This command is available when you right-click a component in the **Console** hierarchy and point to the **Batch** submenu.

If batch processing is configured for the selected item, the command displays the **Schedule Wizard** so you can specify whether the update should perform now or at a scheduled date and time. The update is performed on the Batch Server. For more information, see *Batch Schedule Wizard Common Tasks* (on page 302).

For volume drawings, the **Batch** > **Update Document(s)** command is not available until you place drawing volumes for a volume component in the Space Management task. For composed drawings, this command is not available until you create the drawings in a 3D task. For reports, this command is not available until you create the report by choosing a report template.

To view or modify the currently scheduled batch update jobs, see *Manage batch jobs* (on page 292).

■ NOTES

The software preserves many of the modifications you make between regenerations of volume drawings. For example, if you annotate a volume drawing and then regenerate it, your annotations still display on the updated drawing.

To update a single drawing, right-click a drawing and select **Update** (for marine mode Drawings by Rule) or **Update Now** (for all other drawing types).

 If the software cannot make a SmartPlant Foundation server connection when you use Batch > Update for 3D Model Data documents, you are prompted to provide a valid login and password.

See Also

Updating Documents (on page 82)

Batch Refresh

Allows you to refresh documents on a Batch Server while you continue to work on other tasks. The **Batch > Refresh** command compares the date of the last update of the document with the modification date in the model for any object that has a *positive* (can be seen) resymbolization in the drawing. You can set up batch update jobs for a single document or a select set of documents you select in the **Detail View**. This command is available when you right-click on a single drawing or a select set of drawings in the **Detail View**.

The **Batch > Refresh** command displays the **Schedule Wizard** so you can specify whether the drawing should be updated now or at a scheduled date and time. The batch refresh job runs on a batch server. If the command is not available on the shortcut menu for your documents, batch processing is not configured.

The **Batch > Refresh** command **Schedule Wizard** works the same as the one for the **Batch > Update** command. To walk-through the **Schedule Wizard** and set up batch refreshes, see *Batch Schedule Wizard Common Tasks* (on page 302).

To view and manage current batch jobs, see Manage batch jobs (on page 292).

■ NOTE Batch > Refresh is not available for Spreadsheet Report documents. Spreadsheet Report documents regenerate each time you run, update, or print the report. It is supported for all types of drawing documents.

See Also

Batch Processing (on page 289)
Batch Update Document(s) (on page 300)
Update Now (on page 85)
Update (on page 89) (marine mode Drawings by Rule)

Schedule Wizard

The **Schedule Wizard** displays when you are configured to use batch scheduling, which is available for **Update**, **Refresh**, and **Print**. The **Batch > Update**, **Batch > Refresh**, and **Batch > Print** commands display on the shortcut menu when you select document(s) or components. You can submit an existing batch job request or schedule a new one. You can also multi-select documents within the **Detail View**.

NOTE Batch > Refresh is not available for Spreadsheet Report documents. Spreadsheet Report documents regenerate each time you run, update, or print the report. It is supported for all types of drawing documents.

If you access the Schedule Wizard for a document that already has a batch job scheduled, you can edit or delete the existing batch job. For more information, see *Edit or Delete Batch Jobs* (on page 292).

You can also manage your existing batch jobs with **Tools > Batch Management**. For more information on using batch processing, see *Batch Processing* (on page 289).

To walk-through the **Schedule Wizard** and set up batch updates, see *Batch Schedule Wizard Common Tasks* (on page 302). For batch printing, see *Batch Print Schedule Wizard Common Tasks* (on page 310).

■ NOTE The default timeout value for updating documents through the Batch Server is 40 minutes. For more information on setting the **Batch timeout (in minutes)** property on a drawing document, see *Style Tab (Properties Dialog Box)* (on page 55).

Batch Schedule Wizard Common Tasks

The following Schedule Wizard tasks are used when you schedule batch update or refresh jobs for drawings and reports documents.

■ NOTE Batch > Refresh is not available for Spreadsheet Report documents. Spreadsheet Report documents regenerate each time you run, update, or print the report. It is supported for all types of drawing documents.

The Schedule Wizard displays when you are configured to use a batch server and select **Batch** > **Update** or **Batch** > **Refresh** from the shortcut menu for a selected document(s).

Submitting or Scheduling a Batch Update or Refresh Job

The initial page of the **Schedule Wizard** allows you to specify whether you want to submit a batch update or refresh job now or schedule it for later. For more information, see *Submit or Schedule a Batch Job (Schedule Wizard)* (on page 303).

Setting Batch Job Frequency

If you selected the **Schedule the batch job** option on the initial page of the **Schedule Wizard**, the second page specifies the batch job frequency, or how often you want the batch job to update or refresh. For more information, see *Set Batch Job Frequency (Schedule Wizard)* (on page 305).

Scheduling a Daily Batch Job

When you select the **Daily** option on the second page of the **Schedule Wizard**, you specify the day and time you want the batch job to start. For more information, see *Schedule Daily Batch Job (Schedule Wizard)* (on page 306).

Scheduling a Weekly Batch Job

When you select the **Weekly** option on the second page of the **Schedule Wizard**, you specify the time and day you want the job to start on a per week basis. For more information, see *Schedule Weekly Batch Job (Schedule Wizard)* (on page 307).

Scheduling a Monthly Batch Job

When you select the **Monthly** option on the second page of the **Schedule Wizard**, you specify the time and day you want the batch job to start and the months in which you want the job to run. For more information, see *Schedule Monthly Batch Job (Schedule Wizard)* (on page 308).

Scheduling a One-Time-Only Batch Job

When you select the **One time only** option on the second page of the **Schedule Wizard**, you specify the time and day you want the batch job to start. For more information, see *Schedule One-Time-Only Batch Job (Schedule Wizard)* (on page 309).

Completing the Scheduling

After you have specified the frequency, date, and time settings for your batch job schedule, the final wizard page displays. This page also displays if you selected the **One time only** option on the second page of the wizard. For more information, see *Complete Batch Schedule (Schedule Wizard)* (on page 310).

For documents that have existing batch jobs, the **Schedule Wizard** initial page is different.

Scheduling a New Batch Job

To create a new batch job for the selected document(s), select the **Schedule a new job** option. When you click **Next**, the behavior is the same as the tasks described above, starting with *Set Batch Job Frequency (Schedule Wizard)* (on page 305).

Editing or Deleting Existing Batch Jobs

When you select **Batch > Update** or **Batch > Refresh** on the shortcut menu of a document that already has a scheduled batch job, you can edit or delete an existing batch job by selecting the **Edit existing job(s)** option on the second page of the Schedule Wizard. For more information, see *Edit or Delete Batch Jobs* (on page 292).

Submit or Schedule a Batch Job (Schedule Wizard)

Specifies whether you want to submit a batch update or refresh job now or schedule it for later.



Submit the batch job now - Specifies that the job will be automatically named and submitted to the Batch Server when you click **Finish**. The batch job name defaults using the convention **BatchJob1**, **BatchJob2**, and so forth to keep the batch job names unique. This option is specified by default when you select **Update** from the **Batch** submenu for a document that has no previously scheduled batch jobs.

Schedule the batch job - Specifies that you want to set a date and time for the batch job to run. Click **Next** to go to the next page of the wizard to continue scheduling the batch job. For more information, see *Set Batch Job Frequency (Schedule Wizard)* (on page 305).

■ NOTES

- If the document from which you accessed the **Schedule Wizard** already has batch jobs scheduled, the **Schedule the batch job** option is the default selection. When you click **Next**, you can either create a new batch job for the document or edit an existing one. You can also delete an existing batch job. For more information, see *Update an Existing Batch Job* (Schedule Wizard) (on page 304).
- Batch > Refresh is not available for Spreadsheet Report documents. Spreadsheet Report documents regenerate each time you run, update, or print the report. It is supported for all types of drawing documents.

See Also

Schedule Wizard (on page 301) Updating Documents (on page 82) Batch Processing (on page 289)

Update an Existing Batch Job (Schedule Wizard)

Creates a new batch job for the document, edits the existing batch job, or deletes the existing batch job.

NOTE You can also use **Tools > Batch Management** to update existing batch jobs. For more information, see *Manage batch jobs* (on page 292).



Schedule a new job

Specifies that you are creating a new batch job for the document. Click **Next** to display the next page of the Schedule Wizard and define a new batch job. For more information, see Set Batch Job Frequency (Schedule Wizard) (on page 305).

Edit an existing job

Specifies that you want to edit or delete an existing batch job definition. When you select this option, the table at the bottom of the page enables.

× Delete

Deletes the batch job selected in the table at the bottom of the dialog box. This button is only enabled when a batch job is selected.

Existing batch job table

Lists the batch jobs for the document. To edit a batch job definition, select it in the table and click **Next** to display the next page of the Schedule Wizard. For more information, see *Set Batch Job Frequency (Schedule Wizard)* (on page 305).

See Also

Schedule Wizard (on page 301)
Updating Documents (on page 82)
Batch Processing (on page 289)

Set Batch Job Frequency (Schedule Wizard)

Specifies the frequency with which you want to update or print this document or set of documents. This page of the wizard displays under two different conditions:

- If the document(s) has no previously schedule batch jobs and you select the Schedule the batch job option on the initial page of the Schedule Wizard.
- If the document(s) has previously scheduled batch jobs in existence, and you select the Schedule a new batch job option on the initial page of the Schedule Wizard.



Type a name for this batch job.

Specifies the batch job name. The batch job name defaults using the convention **BatchJob1**, **BatchJob2**, and so forth to keep the batch job name unique. You can change the batch job name.

Perform this job:

Specifies the frequency option for updating the batch job. The options are described as follows:

- Daily The batch job runs daily at the time specified on the next page of the wizard. For more information, see Schedule Daily Batch Job (Schedule Wizard) (on page 306).
- Weekly The batch job runs weekly on the day and time specified on the next page of the wizard. For more information, see Schedule Weekly Batch Job (Schedule Wizard) (on page 307).
- Monthly The batch job runs monthly on the month, day, and time specified on the next page of the wizard. For more information, see Schedule Monthly Batch Job (Schedule Wizard) (on page 308).
- One time only The batch job runs one time only. For more information, see Schedule One-Time-Only Batch Job (Schedule Wizard) (on page 309).

See Also

Schedule Wizard (on page 301)
Updating Documents (on page 82)
Batch Processing (on page 289)

Schedule Daily Batch Job (Schedule Wizard)

Specifies the time and day you want the batch job to start. When you click **Next**, the final wizard page displays the batch schedule settings. If the schedule is incorrect, click **Back** to return to previous wizard pages and make corrections.



Start time

Specifies the time for the batch job to start. You can select a time using the scroll button or define a time in the format shown.

Perform the job

Specifies the day option for updating the batch job. The options are described as follows:

- Every Day The batch job runs every day at the time specified at the top of the wizard page.
- Weekdays The batch job runs every weekday (Monday through Friday) at the time specified at the top of the wizard page.
- Every You specify a number of days in the field provided. The batch job runs once per the specified count of days. For example, if you specified 2 as the value, the batch job runs once every 2 days.

Start date

Specifies the date on which the batch job begins.

See Also

Schedule Wizard (on page 301)
Updating Documents (on page 82)
Batch Processing (on page 289)
Complete Batch Schedule (Schedule Wizard) (on page 310)

Schedule Weekly Batch Job (Schedule Wizard)

Specifies the time and day you want the batch job to start on a weekly basis. When you click **Next**, the final wizard page displays the batch schedule settings. If the schedule is incorrect, click **Back** to return to previous wizard pages and make corrections.



Start time

Specifies the time for the batch job to start. You can select a time using the scroll button or define a time in the format shown.

Every

Specifies a number of weeks. The batch job runs once per the specified count of weeks. For example, if you specified **2** as the value, the batch job will run once every 2 weeks.

Day checkboxes

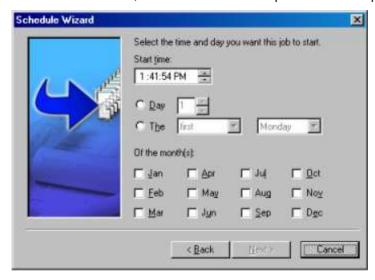
Specifies the day of the week you want the batch job to run.

See Also

Schedule Wizard (on page 301)
Updating Documents (on page 82)
Batch Processing (on page 289)
Complete Batch Schedule (Schedule Wizard) (on page 310)

Schedule Monthly Batch Job (Schedule Wizard)

Specifies the time and day you want the batch job to start and in which months you want the job to run. When you click **Next**, the final wizard page displays the batch schedule settings. If the schedule is incorrect, click **Back** to return to previous wizard pages and make corrections.



Start time

Specifies the time for the batch job to start. You can select a time using the scroll button or define a time in the format shown.

Day

Specifies a specific day of the month. The batch job runs once per the day specified. For example, if you specified **2** as the value, the batch job runs on the second day of the selected months.

The set weekday

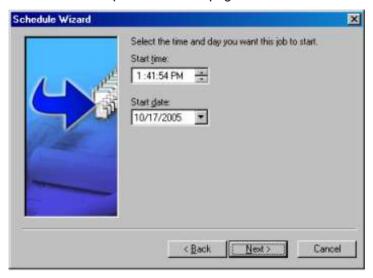
Specifies the day of the week you want the batch job to run. For example, you can set the batch job to run on the **second Tuesday** of every selected month.

Of the month(s)

Specifies the months you want the batch job to run. You can select multiple months.

Schedule One-Time-Only Batch Job (Schedule Wizard)

Specifies the time and day you want the one-time-only batch job to start. When you click **Next**, the final wizard page displays the batch schedule settings. If the schedule is incorrect, click **Back** to return to previous wizard pages and make corrections.



Start time

Specifies the time for the batch job to start. You can select a time using the scroll button or define a time in the format shown.

Start date

Specifies the date on which the batch job update begins.

See Also

Schedule Wizard (on page 301)
Updating Documents (on page 82)
Batch Processing (on page 289)
Complete Batch Schedule (Schedule Wizard) (on page 310)

Complete Batch Schedule (Schedule Wizard)

Shows the completed schedule setup for the batch job. It displays the name of the batch job and the time and date when the job will run.



If the schedule is incorrect, click **Back** to return to previous wizard pages and make corrections. If the batch job is scheduled correctly, click **Finish** to process the batch job request.

See Also

Schedule Wizard (on page 301) Updating Documents (on page 82) Batch Processing (on page 289)

Batch Print Schedule Wizard Common Tasks

The following Schedule Wizard tasks are used when you schedule batch print jobs for drawings and reports documents.

The Schedule Wizard displays when you are configured to use a batch server and select **Batch** > **Print** from the shortcut menu for a selected document(s).

Submitting or Scheduling a Batch Print Job

The initial page of the **Schedule Wizard** allows you to specify whether you want to submit a batch print job now or schedule it for later. For more information, see *Submit or Schedule Printing (Schedule Wizard)* (on page 312).

Setting Batch Job Frequency

If you selected the **Schedule the batch job** option on the initial page of the **Schedule Wizard**, the second page specifies the batch job frequency, or how often you want the batch job to update. For more information, see *Set Batch Job Frequency (Schedule Wizard)* (on page 305).

Scheduling a Daily Batch Job

When you select the **Daily** option on the second page of the **Schedule Wizard**, you specify the day and time you want the batch job to start. For more information, see *Schedule Daily Batch Job (Schedule Wizard)* (on page 306).

Scheduling a Weekly Batch Job

When you select the **Weekly** option on the second page of the **Schedule Wizard**, you specify the time and day you want the job to start on a per week basis. For more information, see *Schedule Weekly Batch Job (Schedule Wizard)* (on page 307).

Scheduling a Monthly Batch Job

When you select the **Monthly** option on the second page of the **Schedule Wizard**, you specify the time and day you want the batch job to start and the months in which you want the job to run. For more information, see *Schedule Monthly Batch Job (Schedule Wizard)* (on page 308).

Scheduling a One-Time-Only Batch Job

When you select the **One time only** option on the second page of the **Schedule Wizard**, you specify the time and day you want the batch job to start. For more information, see *Schedule One-Time-Only Batch Job (Schedule Wizard)* (on page 309).

Completing the Scheduling

After you have specified the frequency, date, and time settings for your batch job schedule, the final wizard page displays. This page also displays if you selected the **One time only** option on the second page of the wizard. For more information, see *Complete Batch Schedule (Schedule Wizard)* (on page 310).

For documents that have existing batch jobs, the **Schedule Wizard** initial page is different.

Scheduling a New Batch Job

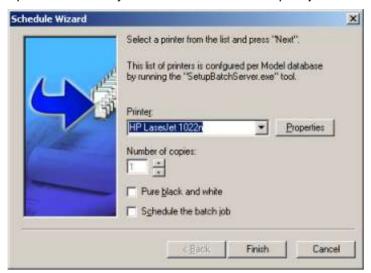
To create a new batch job for the selected document(s), select the **Schedule a new job** option. When you click **Next**, the behavior is the same as the tasks described above, starting with *Set Batch Job Frequency (Schedule Wizard)* (on page 305).

Editing or Deleting Existing Batch Jobs

When you select **Batch > Print** on the shortcut menu of a document that already has a scheduled batch job, you can edit or delete an existing batch job by selecting the **Edit existing job(s)** option on the second page of the Schedule Wizard. For more information, see *Edit or Delete Batch Jobs* (on page 292).

Submit or Schedule Printing (Schedule Wizard)

Specifies whether you want to submit a batch print job now or schedule it for later.



Printer

Specifies the printer to which the document will be submitted.

Number of copies

Indicates the number of copies of the document to print.

Pure black and white

Specifies that the document should be printed in pure black and white, with the software setting the color alterations based on colors used in the document.

Schedule the batch job

Specifies that you want to set a date and time for the batch job to run. Click **Next** to go to the next page of the wizard to continue scheduling the batch job. For more information, see *Set Batch Job Frequency (Schedule Wizard)* (on page 305). If you do not check this box, the document prints immediately to the selected printer.

NOTE If the document from which you accessed the **Schedule Wizard** already has batch jobs scheduled, the **Schedule the batch job** option is the default selection. When you click **Next**, you can either create a new batch job for the document or edit an existing one. You can also delete an existing batch job. For more information, see *Update an Existing Batch Job* (Schedule Wizard) (on page 304).

See Also

Batch Schedule Wizard Common Tasks (on page 302) Schedule Wizard (on page 301) Updating Documents (on page 82)

Custom Commands

Provides you with application programming capability for the 3D software. Using Microsoft® Visual Basic, you can create a custom command that groups a series of commands and instructions into a single command that runs as an operation in the 3D software. As a result, you can access the customized commands that directly relate to the work routine in your operation.

In Visual Basic, the **Command Wizard** helps you to build a custom command. For example, the first **Command Wizard** step prompts you to identify general information, including command name, project name, author, and company. You can start the wizard in Visual Basic by clicking **Command Wizard** on the **Add-Ins** menu. For more information about installing the **Command Wizard** and other programming resources, see the *Intergraph Smart* 3D *Installation Guide* available by clicking **Help > Printable Guides** in the software.

After adding a custom command in the 3D software, you can edit it. The **Edit Custom Command** dialog box requires you to specify the program identifier (prog_id), command name and description, command priority, and a command line of arguments in a string.

Delivered Custom Commands

The following list provides descriptions and ProgIDs for the delivered custom commands:

Custom Command	ProgID	Description
Check Database Integrity	SP3DCheckDatabaseIntegrity . CCheckObj	Creates records for the objects that need to be cleaned. You run this custom command directly on a database (Site, Catalog, or Model). After you run this command, you can generate a report to review the errors that the Check Database Integrity command generated. For more information on the Check Database Integrity command, see the Database Integrity Guide available from Help > Printable Guides.
Clean Database	SP3DCleanDatabaseCmd. CCheckObj	Deletes or cleans an object. This command is used when an action on the Check Database Integrity report is To Be Removed or To Be Repaired. For more information on deleting and cleaning objects in the database, see the Database Integrity Guide available from Help > Printable Guides.

Custom Command	ProgID	Description
Create Drawing View	MenuDrawView. CMenuDrawView	Saves and converts the contents of a three- dimensional graphic view window into a snapshot view. The command creates a rectangular object associated to a clipping volume or volumes in the three-dimensional model.
		Before you create a snapshot view using this command, you must have added at least one composed drawing type to the Management Console in the Drawings and Reports task.
		You can save additional views by updating the view contents and then saving the new design. If you used the Tools > Hide command to avoid displaying certain objects, those objects are included in a composed drawing you create.
		You must have appropriate permissions to access composed drawing types, or you cannot use the Tools > Snapshot View command. If you have only read permission, you receive a message that alerts you to this condition.
		After you create the snapshot views, you can add them to composed drawings when you use the Tools > Drawings Console command.
Drawings Check and Repair Utility	DwgCheckUtility,Ingr.SP3D. Drawings.Client.Commands. DwgCheckUtility. RunChecksCmd	Checks drawing items for problems, such as mismatches between views, smartframes, and OIDs; duplications of views, smartframes, and OIDs; and invalid dimensions with missing smartframe attributes. After problems are found, you can run repairs.
		This command is intended for use by your administrator.

Custom Command	ProgID	Description
Find Object by OID	SP3DFindObjectByReport. FindObjects	Finds objects with integrity problems in a graphic view. Before running this command, you must define your workspace to include these objects. Run a database integrity report, and use the reported OIDs of the objects in the workspace definition. For more information on the Find Objects by OID custom command, see the <i>Database Integrity Guide</i> available from Help > Printable Guides .
Fix Project Root	SP3DPRJMGTRepairCmd. FixCnfgProjectRoot	Synchronizes the model name in the Model database and the Site database. The name in the Site database prevails.
		NOTE You must run this command from a task in the model, not from Project Management.
Fix Sector Size of Documents	DwgBinaryEditorCmd. FixSectorSize	Adjusts the sector size from small to large. Documents that have many sheets and a small sector size can cause the software to run out of memory.
Reset Design Basis Time	IMSEngFrameworkCmd. EngFrameworkCmd Argument = ResetDesignBasisTime	Modifies the Design Basis timestamp. With this command, you can set the time and date back to a point in the past. This command is useful if, for any reason, there are delete instructions that could not be processed.
Synchronize Drawing Component Templates	DwgSynchTemplatesCmd.Sy nchTemplates	Repairs a drawing component that has become corrupted by synchronizing it with a different, uncorrupted drawing component. This command requires that the source component is the same type as the corrupted component, the source component must have a template, and that the source component cannot be corrupted. Also, the number of views on the source component must be the same as the number of views on the corrupted component. The names of the views on the source component must match the view names of the corrupted component.

Custom Command	ProgID	Description
Verify P&ID Integrity	SP3DDisplayPIDService. VerifyPIDCmd	Validates the internal connections between objects on a P&ID and objects in the Model database. This command is useful when there is a problem displaying a P&ID or selecting objects on a P&ID. The command provides some basic troubleshooting statistics: Number of design basis objects, number of 3D objects (correlated), number of P&ID objects, number of deleted P&ID OIDs, and number of duplicate OIDs.

Custom Commands Dialog Box (on page 318)

Add Custom Command Dialog Box (on page 318)

Edit Custom Command Dialog Box (on page 319)

What do you want to do?

- Create custom commands (on page 316)
- Add custom commands (on page 317)
- Run a custom command (on page 317)
- Edit a custom command (on page 317)
- Delete a custom command (on page 317)

Create custom commands

1. Open Microsoft® Visual Basic.

TIPS

- You do not create or modify custom commands within the software. You can edit the code of the command in Visual Basic. You can edit a limited number of items, such as the description of the command, using the Edit Custom Command dialog box.
- You must install the Command Wizard software in Visual Basic. The setup for the Command Wizard is located at [Product Folder]\CommonApp\Tools\CommandWizard.
- 2. In Visual Basic, click Add-Ins > Command Wizard.
- 3. Complete all steps on each page of the **Command Wizard**.

Add custom commands

- 1 Click Tools > Custom Commands
- 2. On the Custom Commands dialog box, click Add.
- 3. On the **Add Custom Command** dialog box, type the program identifier you assigned to the command in Microsoft® Visual Basic in the **Command ProgID** box.
- 4. Type the name you assigned to the command in the **Command name** box.
- 5. Type a phrase that describes the command in the **Description** box.
- 6. If necessary, change the option in the **Priority** section.
- 7. Type command line arguments in a string in the **Argument** box.
- TIP After you complete this procedure, the **Custom Commands** dialog box lists the command you added to the software. You can run the command, edit the settings, or delete the command.

Run a custom command

- 1. Click Tools > Custom Commands.
 - The Custom Commands dialog box opens.
- 2. To start a custom command you created, select the command in the list box, and click Run.
- 3. After the command runs, click Close on the Custom Commands dialog box.

Edit a custom command

- 1. Click Tools > Custom Commands.
 - The **Custom Commands** dialog box opens.
- 2. To change the options for a custom command, select the command in the list box, and click **Edit**. For example, you can change the name and description of the command.
- 3. After completing the needed changes, click **Close** on the **Custom Commands** dialog box.
- **NOTE** You must open the command in Microsoft® Visual Basic if you want to edit the underlying code.

Delete a custom command

- 1. Click Tools > Custom Commands.
 - The Custom Commands dialog box opens.
- 2. Select the command in the list box, and click **Delete**. The software removes the command from the list box; however, the command code is not deleted.
- 3. After completing the needed changes, click Close on the Custom Commands dialog box.
- **NOTE** This action does not delete the DLL for the custom command. It just removes access to the custom command from the **Custom Commands** dialog box.

Custom Commands Dialog Box

Adds and edits customized commands you have created with the **Command Wizard** in Microsoft® Visual Basic. For information on creating custom commands, see *Create custom commands* (on page 316).

Command names

Lists the names of commands that have been added.

Run

Starts the custom command you select in the list box. For more information, see *Run a custom command* (on page 317).

Close

Cancels the Custom Commands dialog box.

Edit

Opens the **Edit Custom Command** dialog box. You can change settings for the command, such as the program identifier (prog_ID) and command name. For more information, see *Edit a custom command* (on page 317).

Add

Installs the custom command into the software. For more information, see *Add custom commands* (on page 317).

Delete

Removes the custom command from the software. For more information, see *Delete a custom command* (on page 317).

Clear

Deletes the information you have typed in the boxes on the **Custom Commands** dialog box.

Description

Contains an identifying phrase so you can better recognize the custom command with which you are working.

Add Custom Command Dialog Box

Accesses a customized command you created in Microsoft® Visual Basic and saves the command within the software.

Command ProgID

Identifies the program identifier for the custom command you created in Visual Basic.

Command name

Specifies the name you assigned to the custom command.

Description

Describes the custom command.

Priority

Assigns a priority of High, Normal, or Low.

Argument

Specifies command line arguments in a string.

See Also

Create custom commands (on page 316)

Edit Custom Command Dialog Box

Changes options for a customized command you added to the software.

Command ProgID

Specifies the program identifier for the custom command you created in Microsoft® Visual Basic.

Command name

Provides a text box for you to change the name you assigned to the custom command.

Description

Provides a text box to provide a descriptive phrase for the custom command.

Priority

Changes priority to **High**, **Normal**, or **Low**.

Argument

Change the command line arguments in a string.

Reset Default

Returns the dialog box to its default settings.

See Also

Create custom commands (on page 316)

Delivered Custom Commands

This section describes some of the delivered custom commands. For a comprehensive list of the custom commands, see *Custom Commands* in the *Common User's Guide*.

Repair Documents Custom Command

The **Repair Documents** custom command updates invalid **Styles.sha** or **Symbol Browser** file paths on a document or a set of documents. Invalid file paths can occur when you change the symbol share. This causes overhead while editing, saving, or updating documents.

Click **Tools > Custom Commands** to add the command using the ProgID **DwgRepairCmd.RepairDocuments**. For more information, see *Custom Commands* in the *Common User's Guide*.

Each time you use **Repair Documents**, the changes made to the component or document are saved in a log file. You can access the file at %TEMP%\Drawings. The log file name starts with "DwgRepairCmd_" as its prefix. A summary at the end of the log file lists all documents that the command could not repair.

Repair Documents Dialog Box

Component or Document

Specifies the component or document to repair.

Actions

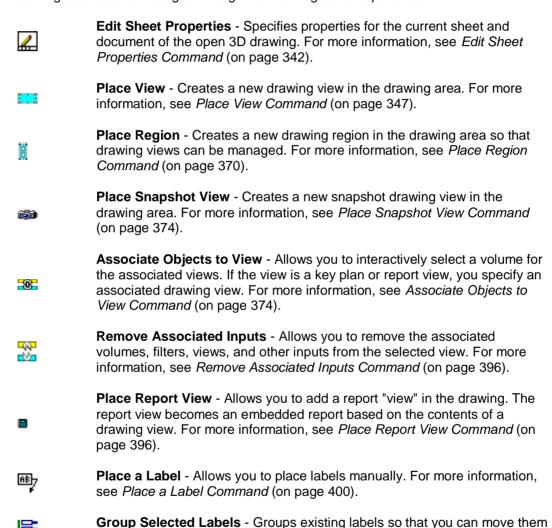
Shows the repair options for the selected component or document.

- Reset style resources Resets the style resource files and to the Styles.sha file on the current symbols share.
- Set symbol browser home Sets the symbol browser home address. You can change
 this address by typing in a new address or by clicking More and opening the file
 folder.

SECTION 24

Working with Drawings and Reports and **SmartSketch Drawing Editor**

The following commands are available when you are using SmartSketch Drawing Editor to edit a drawing created and managed through the Drawings and Reports task.



2D/3D Selection - Allows you to move back and forth between the SmartSketch Drawing Editor application window and the 3D task window. For more information, see 2D/3D Selection Command (on page 443).

as a unit. For more information, see *Group Selected Labels* (on page 419).

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Scaled Sketching - Enables you to edit or draw new objects at a scale that is different from the drawing scale for the active sheet. For more information, see *Scaled Sketching Command* (on page 446).



Edit Border Family - Associates a single border or family of borders to the current layout template. For more information, see *Edit Border Family Command* (on page 483).



Hide/Show Object - Hides or displays objects in the drawing view. For more information, see *Hide/Show Object Command* (on page 455).



Copy and Paste View - Copies an orthographic drawing view and places the copy on the same sheet. For more information, see *Copy and Paste View Command* (on page 458).



Move View - Moves one or more views from a composed drawing document to another composed drawing document. For more information, see *Move View Command* (on page 458).



Highlight Annotations - Highlights labels, dimensions, and customized graphics based on the options that you select. For more information, see *Highlight Annotations Command* (on page 460).



Clear Manual Edits - Permanently clears all manual edits made to labels and dimensions in the selected views. For more information, see *Clear Manual Edits Command* (on page 462).



Associate Objects to Graphic View - Associates and disassociates objects to graphic views. For more information, see *Associate Graphics to Graphic View Command* (on page 463).

Right-click menu for selected views

Update View - Updates the contents for the selected view - drawing, report, key plan, snapshot, and detail/section views. For more information, see *Update View Command* (on page 485).



Preview Layout - Shows a preview of the drawing document based on the current layout. For more information, see *Preview Layout Command* (on page 484).



Place Drawing View - Places a two-dimensional view for orthographic drawings. For more information, see *Place Drawing View Command (Template Toolbar)* (on page 466).



Place Report - Embeds a report in a drawing view on a volume or snapshot drawing. For more information, see *Place Report Command (Template Toolbar)* (on page 469).



Place Key Plan - Places a key plan on a volume drawing template. For more information, see *Place Key Plan Command (Template Toolbar)* (on page 472).



Place Border Label - Positions drawing property labels in the title block of a template when you are editing a 3D drawing border template. For more information, see *Place Drawing Property Label Command (Drawing Labels Toolbar)* (on page 475).



Place Drawing Area - Places a drawing area. You use this command to create a drawing area on a 3D drawing border template that has been imported from other software, such as MicroStation DGN or AutoCAD DWG. For information on this command, see *Place Drawing Area Command* (on page 480). For more information on editing border templates, see the *Drawings and Reports User's Guide*.

Drawings View Explorer

The Drawings View Explorer provides access to views and their attributes for the currently displayed drawing or template. For more information, see *Drawings View Explorer* (on page 324).

Dimensioning in 3D Drawings

You can use the Dimension toolbar to place dimensions on your 3D drawings. For more information, see *Dimensioning in 3D Drawings* (on page 337).

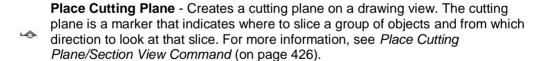
You can also specify dimensioning for paper space objects (at the proper scale) when they are drawn on top of drawing view objects. For more information, see *Dimension Paper Space Objects for 3D Drawings* (on page 338).

Cutting Planes, Detail Envelopes, and Section/Detail Views

The following commands allow you to create and edit cutting planes, detail envelopes, and section and detail views in 3D drawings.



Place Detail Envelope - Creates a detail view for an existing drawing view. Detail views are more than enlargements of the main drawing view. They often contain additional graphical information that is not visible in the main drawing view, such as weld or chalk information. For more information, see *Place Detail Envelope Command* (on page 423).





Place Section/Detail View - Creates both a section view and a detail view based on the selected cutting plane or detail envelope. A detail view is extracted from a main drawing view or another detail view. You can rotate detail views in 2D space, but they remain in the same orientation as the main drawing view. Section views are extracted from main drawing views, detail views, or other section views. Section views are similar to detail views, except that they can display information in an orientation that is different from that of the originating view. For more information, see *Place Detail View Command* (on page 439).

Converting Excel Spreadsheet Reports to Native Text Boxes

The **SP3DConvertExcelEmbedded.dll** is a delivered custom command that allows you to convert an Excel spreadsheet report to the native text box format for use in 3D Drawings. For more information on converting Excel spreadsheet reports, see *Convert Excel Spreadsheet Reports to Native Text Box Format Custom Command* (on page 491).

Saving Drawings in MicroStation and AutoCAD Format

You can use the Save As command to assign dimension units and export the drawings to MicroStation or AutoCAD format. For more information, see *Save As MicroStation or AutoCAD Format* (on page 340).

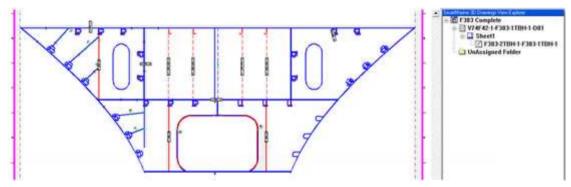
Drawings View Explorer

Provides access to views and their attributes for the currently displayed drawing or template.

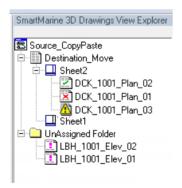
Drawings View Explorer

Displays a list of sheets and views on the active sheet and a list of unplaced views available for placement. This window is available in SmartSketch Drawing Editor when you have a Drawings by Rule template or drawing open. The **Drawings View Explorer** displays by default as a right-side window in SmartSketch Drawing Editor when you:

- Edit a template associated with a Drawings by Rule component. Right-click the component, and select **Edit Template** on the shortcut menu.
- Edit an existing drawing document. Right-click a Drawings by Rule drawing document in the Detail View or the Drawing Console, and select Edit on the shortcut menu.



The **Drawings View Explorer** allows you to see views that are available for the current drawing. Assigned views appear beneath a drawing sheet. Unassigned views appear in the **UnAssigned Folder**. Views are identified by the following icons:



- The view is assigned to a drawing and all 2D geometry is up-to-date with the 3D model.
- The view is assigned to a drawing and 2D geometry is out-of-date with the 3D model.
- The view is not assigned to a drawing, and is in the **UnAssigned Folder**.

- The view is assigned to a drawing, and there were errors in processing the 2D geometry. The update skips the geometry errors.

Shortcut Menu

The **Drawings View Explorer** also allows you to update, refresh, delete, or edit properties by right-clicking on a view or a set of selected views from the same sheet.

Update

Performs a smart update of geometry in the view(s). Rules in the software determine whether to perform:

- An incremental update of the geometry for added, modified, and deleted objects.
- A full update of all geometry.

For more information, see Update and Full Update Commands (on page 504).

NOTE When **Update** performs the first option, it is usually faster than **Full Update** but still results in all geometry being up-to-date for the view.

Full Update

Performs a full update of all geometry in the view. For more information, see *Update and Full Update Commands (on page 504).*

Refresh

Compares the date of the last update of the views with the modification date in the model for any object that has a *positive* (can be seen) resymbolization in the drawing. For more information, see *Refresh* in the *Orthographic Drawings User's Guide*.

Delete

Moves one or more views to the **UnAssigned Folder**, or deletes views permanently if the view was placed by the drawing rule set. For more information, see *Place View Command* (on page 347).

Properties

Opens the **Drawing View Properties** dialog box, allowing you to change the properties of a drawing view. For more information, see *Drawing View Properties Dialog Box (Place View Command) - Steel Order Drawings* (on page 354).

Update Selected

Performs an update on objects that are first selected in a view or in the 3D model. You can perform **Update Selected** on both full and cropped views. Only the selected geometry is updated in the view. Other geometry affected by the selected geometry is neither checked

nor updated. As a result, the view is marked as out-of-date . This option is only available when:

- A drawing is created with a Drawings by Rule component. For more information, see the Orthographic Drawings User's Guide.
- 2D/3D Selection se

View Log

Displays a log of the results from the last update performed on the view.

■ NOTES

- You can also select multiple views from the RAD sheet.
- For more information, see Place an Unassigned View (on page 497).

Move Sheet(s)

Moves the selected sheets from one document to another under the same Drawings by Rule component. For more information, see *Move Sheet(s)* (on page 327).

Move Sheet(s)

Moves the selected sheets from one document to another under the same Drawings by Rule component.

1. In the Drawings View Explorer, right-click the sheets to be moved, and select **Move Sheet(s)**.

The Move Sheet dialog box displays.

2. Select the destination document, and click **OK**.

NOTE Sheets cannot be moved to a document that is being edited or to a document on which you do not have write permissions.

The sheets are moved to the destination document.

■ NOTES

- You can only move sheets to documents under the active component.
- When all sheets under a document are moved, an empty sheet with a border template is placed under the source document.
- To perform Move Sheet(s), the component must have at least two documents as children.
- Move Sheet(s) cannot be used by selecting both views and sheets together.
- Upon completion, the status of the source and destination documents is changed to "Out of date."

Dimensions

In general, dimension rules control the placement of dimensions, and dimension styles control the appearance, including the units, of dimensions in orthographic drawings. However, dimension styles and dimension rules interact in complex ways. There are two methods you can use to cause dimensions to display in drawings. *Automatic dimensioning* and *manual dimensioning* place dimensions in native format drawings.

For *automatic dimensioning*, the view style controls whether or not dimensions are placed. For *manual dimensioning*, you edit an existing drawing and place dimensions manually.

NOTE The **Save As** command saves drawings from the database to files and presents additional considerations about dimensions. The files created by the **Save As** command can be native format or a foreign format such as MicroStation or AutoCAD. With the **Save As** command, the software attempts to replicate dimensions as they are shown within a drawing.

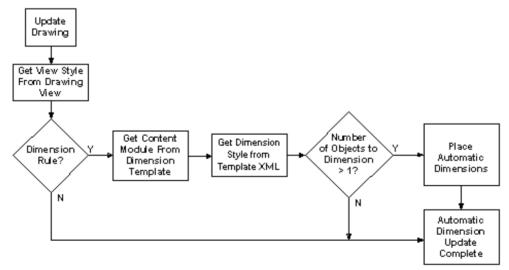
Assigning dimension units is different for each method. For more information, see the following sections in the Orthographic Drawings User's Guide:

Automatic Dimensioning (on page 328)
Manual Dimensioning (on page 335)
Save As MicroStation or AutoCAD Format (on page 340)

■ NOTE Isometric drawings use the isometric options settings within their style to determine the dimension appearance, placement, and units. For more information, see *Isometric Drawing Styles* (on page 125) in the *Isogen Isometric Drawings User's Guide*.

Automatic Dimensioning

When you use automatic dimensioning, the view style controls whether or not dimensions are placed. The following flowchart shows how **Update** (for marine mode Drawings by Rule) and **Update Now** (for all other drawing types) assign units to automatically placed dimensions.



View Styles

A dimension rule within a view style triggers the automatic placement of dimensions in an orthographic drawing. For more information, see *Use View Styles with Dimension Rules* (on page 329).

Dimension Rules

Dimension rules control the placement and appearance of automatic dimensions in the drawing. For more information, see *Use Dimension Rules* (on page 330).

Dimension Templates

The dimension rule is not responsible for assigning the dimension units to automatic dimensions. The dimension rule points to a dimension template XML file that influences the unit assignment. For more information, see *Use Dimension Templates* (on page 331).

Dimension Styles

You can determine the active style of a dimension by editing the drawing or drawing template in **SmartSketch Drawing Editor**. For more information, see *Edit Dimension Styles* (on page 332).

Dimension Style Overrides

You can override dimension style settings in **SmartSketch Drawing Editor**. For more information, see *Override Dimension Styles* (on page 334).

See Also

Dimensions (on page 328)

Use View Styles with Dimension Rules

A dimension rule within a view style triggers the automatic placement of dimensions in a drawing.

- Select Tools > Define View Style.
 - The **Define View Style** dialog box displays.
- 2. Select **New Style** to define a new view style with a dimension rule. You can also edit an existing view style by selecting **Properties**.

3. On the **View Style Properties** dialog box, use the **Dimension Rule** field dropdown to specify a dimension rule for the view style. For more information on this dialog box, see *View Style Properties Dialog Box* in the *Drawings and Reports Reference Data Guide*. Select **More** to display the **Select Dimension Rule** dialog box. For more information, see *Select Dimension Rule Dialog Box* in the *Drawings and Reports Reference Data Guide*.



For information on how dimension rules are maintained, see *Use Dimension Rules* (on page 330).

▶ NOTE For more information on defining view styles, see *Define View Style Command* in the *Drawings and Reports Reference Data Guide*, accessible using the **Help > Printable Guides** command in the Drawings and Reports task.

See Also

Automatic Dimensioning (on page 328) Dimensions (on page 328)

Use Dimension Rules

A dimension rule controls the placement and appearance of automatic dimension in the drawing. When you select **More** in the **Dimension Rule** dropdown on the **View Style Properties** dialog box, the **Select Dimension Rule** dialog box displays the list of rules available in the Drawings catalog. For more information, see *Select Dimension Rule Dialog Box* in the *Drawings and Reports Reference Data Guide*. You can access this guide with the **Help > Printable Guides** command in the Drawings and Reports task.

The Drawings catalog is file-based and located on the SharedContent share within the \\Drawings\\Catalog\\Formulate\Drawings\\Catalog\\Rules\DimensionRules\Text{ files stored in the }\\Drawings\\Catalog\\Rules\DimensionRules\Text{ folder. Several example dimension rules are delivered with the software.}

You edit the dimension rule XML files with a text or XML editor. Each dimension rule should point to its own dimension template. Rename any customized dimension rules. Do not use the delivered rule names for customized rules.

★ IMPORTANT

- We recommend that you maintain a separate set of dimension rules for each dimension style used in drawings.
- The dimension rule is not responsible for assigning the dimension units to automatic dimensions. However, the dimension rule points to another XML file, called the *dimension* template, that influences the unit assignment. For more information, see *Use Dimension* Templates (on page 331).

■ The drawing dimension XML Files are discussed further in the *Intergraph Smart*TM 3D *Programmer's Guide* under *Extending the Capabilities of the Software*. Contact your administrator or Intergraph Support if you need the *Intergraph Smart*TM 3D *Programmer's Guide*. You can find support information on our web site *http://support.intergraph.com* (*http://support.intergraph.com*).

See Also

Automatic Dimensioning (on page 328)
Dimensions (on page 328)

Use Dimension Templates

The dimension rule is not responsible for assigning the dimension units to automatic dimensions. However, the dimension rule points to another XML file, called the *dimension template*, that influences the unit assignment. The dimension template XML file contains the settings that further control the placement and appearance of dimension in the orthographic drawings.

You edit the dimension template XML files with a text or XML editor. Rename any customized dimension templates. Do not use the delivered rule names for customized dimension templates.

Several example dimension template XML files are delivered with the software and are located on the SharedContent share in the \(\mathbb{D}\)rawings\(\mathbb{C}\)atalog\(\mathbb{D}\)imensions\(\mathbb{T}\)emplates folder.

The setting in the template that influences dimension unit display is <dimensionContentModules>. In each dimension template, you should set <dimensionContentModules> to the dimension style used to create dimensions in the drawing. The values available for this setting determine whether the software places the dimension vertical, horizontal, radial, or angular. To determine the dimension units, the various content modules refer to the dimension formatting saved either in the drawing template file (for volume and by query drawing types) or in the drawing itself (for composed drawings).

★ IMPORTANT

- We recommend that you maintain a separate dimension template for each dimension rule you create.
- For manual dimensioning, the software looks in the Linear_A_HV dimension template for the value of the dimension content module during the update of manually placed dimensions. For more information, see Manual Dimensioning (on page 335).
- The <dimensionStyleSettings> setting in the dimension templates is not used by the software at this time. However, when creating drawings, we recommend that the active dimension style in the selected border template have the same name as the <dimensionStyleSettings> setting in the dimension template to avoid problems in future software releases. Rename any customized border templates. Do not use the delivered border template name for customized border templates.
- The drawing dimension XML Files are discussed further in the *Intergraph Smart*TM 3D *Programmer's Guide* under *Extending the Capabilities of the Software*. Contact your administrator or Intergraph Support if you need the *Intergraph Smart*TM 3D *Programmer's Guide*. You can find support information on our web site *http://support.intergraph.com(http://support.intergraph.com/)*.

Overall Dimensions

By default, dimension control generators use the settings in the Linear_A_HV.xml dimension template. By using **<overallDimension>**, you can override Linear_A_HV.xml and select another template for overall dimensions. In the example below, Piping Plan_Pipes_Horizontal.xml is used as the dimension template for piping parts:

```
<dimensionSettings>
     <overall>-1</overall>
     <overallDimension value="Piping Plan_Pipes_Horizontal">
</dimensionSettings>
```

Layering Dimensions

You can define the layer on which your dimensions display in the drawing by editing the dimension rule template XML file. After the **<dimensionStyleSettings>** section of the XML file, add a **<dimensionLayerSettings>** definition using the **<dimLayer>** tag, like the one shown in the example below, that defines the layer on which you want the dimensions placed:

```
<dimensionLayerSettings>
    <dimLayer>dimensionLayer</dimLayer>
</dimensionLayerSettings>
```

If the layer does not already exist in the template you are using to create drawings, the software creates the layer automatically.

See Also

Automatic Dimensioning (on page 328) Dimensions (on page 328) Use Dimension Rules (on page 330)

Edit Dimension Styles

The software resolves dimension units for a particular drawing from the active dimension formatting in either the drawing template (for volume and *by query* drawing types) or the drawing itself (for composed drawings). The active dimension formatting is a combination of the settings in the active dimension style plus any dimension style overrides that may be set.

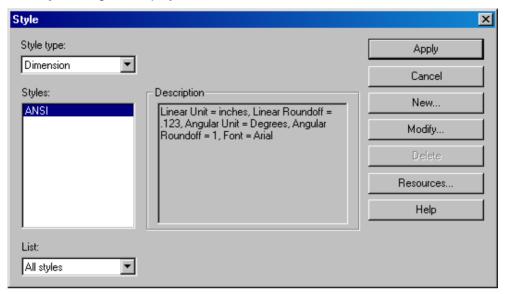
You can determine the active style by editing the drawing or the drawing template in **SmartSketch Drawing Editor**, selecting one of the placement dimension commands, and checking the style that displays on the resulting ribbon bar. You can view the overrides to the active dimension in the drawing template by selecting **Format > Dimension** in **SmartSketch Drawing Editor**.

★ IMPORTANT The Imperial border templates have ANSI as the active dimension style with no overrides set. The Metric border templates have DIN as the active dimension style with no overrides set.

You can create and maintain dimension styles in SmartSketch Drawing Editor.

1. In SmartSketch Drawing Editor, select Format > Style.

The Style dialog box displays.



2. Click Modify.

The Modify Dimension Style dialog box displays.

- 3. Modify the appearance of the dimension style as needed, and click **OK**.
- 4. On the **Style** dialog box, you can either save dimension styles with the files being edited or reference them from other files. You can reference files by selecting **Resources**.

TIPS

- The reference files, also called resource files, can be any file with an .igr or .sha extension. While referenced dimension styles are available for dimension placement, they cannot be edited. Therefore, the Style dialog box only displays dimension styles that are saved locally to the file.
- In cases where a local dimension style has the same name as a referenced dimension style, the software always uses the local style when placing dimensions. Currently, you cannot override local dimension styles with the same name as a referenced dimension style of the same name. Local dimension styles can be renamed so that the referenced dimension style can be used to place dimensions. Local dimension styles are renamed on the **Modify Dimension Style** dialog box. For more information on overriding dimension style settings, see *Override Dimension Styles* (on page 334).

The delivered *Styles.sha* file contains several example dimension styles. This file is located in the Symbols share in the *\Drawings\Catalog\Templates* folder. All drawings created in the Drawings and Reports task reference the *Styles.sha* file.

★ IMPORTANT After updating the drawings, the *Styles.sha* file on the Symbols share is the only file referenced into the drawings. This is true for all orthographic drawings. Any other files previously referenced before the update need to be referenced to the drawing again.

All drawings and drawing templates must have at least one dimension style saved within them. Therefore, if only one dimension style is saved with a file, you cannot delete it. Also, you cannot delete any style from a drawing or drawing template if they are currently used by dimensions in a drawing.

Dimension styles can be added to the list of saved or local styles in a drawing or drawing template in one of two ways:

- Create a new dimension style from the Style dialog box using the New... command.
- Place a dimension in a drawing or drawing template using a reference dimension style. This
 method copies the referenced dimension style into the drawing or drawing template.

★ IMPORTANT

- We recommend that you maintain a separate set of border templates for each dimension style used for drawings. The border templates within a particular set should have the same active dimension style with the same style settings.
- The <dimensionStyleSettings> setting in the dimension templates is not used by the software at this time. However, when creating drawings, we recommend that the active dimension style in the selected border template have the same name as the <dimensionStyleSettings> setting in the dimension template to avoid problems in future software releases. Rename any customized border templates. Do not use the delivered border template name for customized border templates.
- We recommend that you maintain a complete set of your dimension styles in a renamed version of the Styles.sha file.

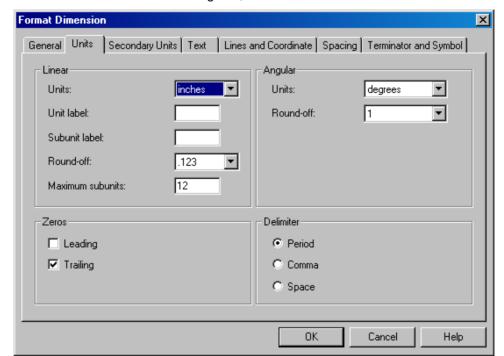
See Also

Override Dimension Styles (on page 334)

Override Dimension Styles

You can override dimension styles in a drawing or drawing template.

- ★ IMPORTANT We support dimension style overrides, but we do not recommend their use as a standard practice. Overrides are saved in the drawing or drawing template and can be difficult to maintain as dimension styles change. As an alternative, we recommend creating a new dimension style in a reference resource file. For more information, see *Edit Dimension Styles* (on page 332).
- 1. Select **Tools > Edit Border Template**, and select a template to edit.
 - The template opens in SmartSketch Drawing Editor.
- 2. Select Format > Dimension.



3. On the Format Dimension dialog box, select the Units tab.

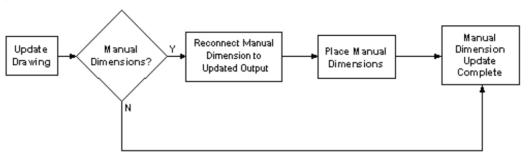
- 4. If the settings on the **Units** tab are identical to those on the **Modify Dimension Style** dialog box, no overrides are applied to the active dimension style. If a dimension style is changed during a dimension placement command, the override values become identical to the dimension style values. For more information, see *Edit Dimension Styles* (on page 332).
- **NOTE** You cannot override local dimension styles with the same name as a referenced dimension style. Local dimension styles can be renamed so that the referenced dimension style can be used to place dimensions. Local dimension styles are renamed on the **Modify Dimension Style** dialog box.

See Also

Automatic Dimensioning (on page 328) Dimensions (on page 328) Use Dimension Rules (on page 330)

Manual Dimensioning

When you use *manual dimensioning*, the view style and the dimension rules do not apply for display or placement. However, the logic used to determine manual dimensions is similar to that of *automatic dimensions*. The following flowchart shows how **Update** (for marine mode Drawings by Rule) and **Update Now** (for all other drawing types) assign units to manually placed dimensions.



Dimension templates

For *manual dimensioning*, the software looks in the dimension template for the value of the dimension content module during the update of manually placed dimensions. For more information, see *Use Dimension Templates* (on page 331).

Dimension styles

You can determine the active style of a dimension by editing the drawing or drawing template in **SmartSketch Drawing Editor**. For more information, see *Edit Dimension Styles* (on page 332).

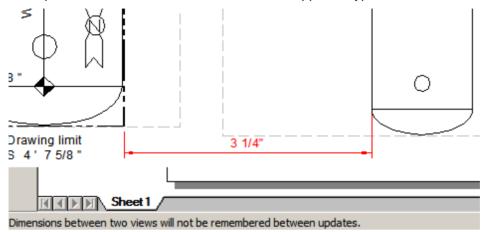
Dimension style overrides

You can override dimension style settings in **SmartSketch Drawing Editor**. For more information, see *Override Dimension Styles* (on page 334).

NOTE If you place a manual linear dimension that is chained or stacked with an automatic linear dimension, the manual dimension is immediately replaced with a standalone dimension that is no longer attached to the automatic dimension. The new dimension is attached to a newly-placed point object whose position coincides with the connect point location of the automatic dimension projection line to which the manual dimension was originally attached. Because the new dimension is no longer chained or stacked with the automatic dimension, it does not move if the automatic dimension's position is modified.

Dimension between views

If you try to place dimensions from one drawing view to another, a status bar message displays **Dimensions between two views will not be remembered between updates.** The dimension is not placed and the command cancels. Other supported types of dimensions remain.



Dimensioning in 3D Drawings

When you edit a Smart 3D drawing in SmartSketch Drawing Editor, you can use commands on the **Dimension** toolbar to manually place dimensions. When updating the drawing, the software remembers dimensions placed between objects within a single drawing view, dimensions placed from paper space to paper space, and dimensions placed between paper space graphics and model objects. You cannot dimension from one drawing view to another drawing view.

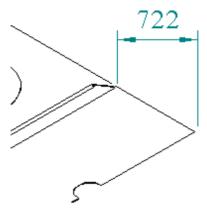
You can place dimensions in drawings by rule and composed drawing views using the standard SmartSketch Drawing Editor dimensioning commands. The commands are enhanced with an additional ribbon to support accurate dimensions of 3D objects in non-planar views.

Dimensioning Drawing Elements in the SmartSketch Drawing Editor Help

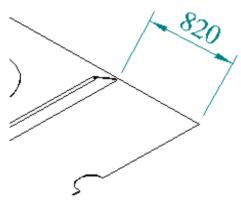
Dimension Ribbon in the SmartSketch Drawing Editor Help

3D Dimension Ribbon

The 2D dimension commands treat all geometry in a view as being in the view plane. If the view contains non-planar geometry, such as in an isometric view, the value of the dimension is not accurate, as shown in the following figure.



With the 3D dimension options, the dimension reflects the distance between the 3D objects.



You can specify dimensioning for paper space objects (at the proper scale) when they are drawn on top of drawing view objects. For more information, see *Dimension Paper Space Objects for 3D Drawings* (on page 338).

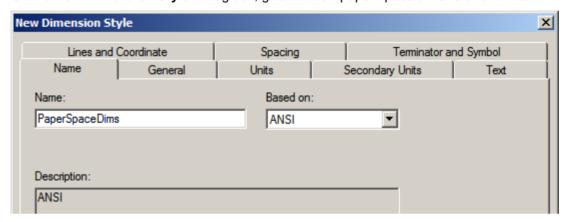
Dimension Paper Space Objects for 3D Drawings

The following procedure shows you how to dimension paper space objects (at the proper scale) when they are drawn on top of a 3D Drawing view object. To accomplish this task, you create a dimension style with the appropriate scale setting in SmartSketch Drawing Editor.

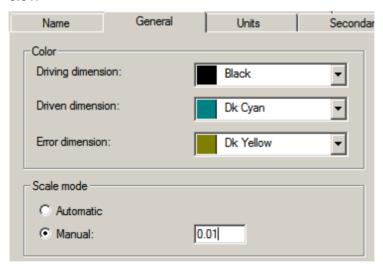
When manually placing dimensions and then updating the drawing, the software remembers dimensions placed between objects within a single drawing view, dimensions placed from paper space to paper space, and dimensions placed between paper space graphics and model objects.

★ IMPORTANT Dimension styles must be defined for paper space objects and their scale factors, otherwise the paper space dimensions will be deleted when the drawing is updated. SmartSketch Drawing Editor will not recognize any possible errors in the dimension style, so be sure that any new dimension styles are correct before updating a drawing. For more information, see Edit Dimension Styles (on page 332) and Manual Dimensioning (on page 335).

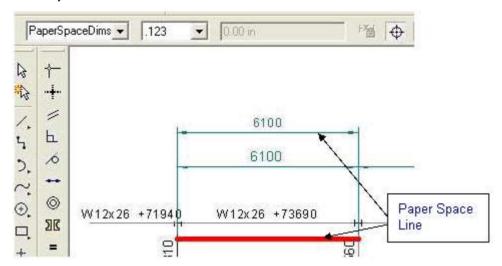
- 1. Start SmartSketch Drawing Editor. Double-click shape2dserver.exe in the [Product Folder]\Common2D\Shape2D\Bin folder.
- 2. Use **File > Open** to open the *styles.sha* file. This file is located in your [*Product Folder*]\Common2D\Shape2D\Template\Styles folder.
 - **NOTE** The *styles.sha* file is delivered as a read-only file. You can change the properties of the file, or you can copy the file to a new name and use the copied file to create your new dimensioning style.
- 3. Select Format > Style, and on the Style dialog box, click New.
- 4. On the **New Dimension Style** dialog box, give the new paper space dimension a name.



5. Go to the **General** tab, and set the scale mode to **Manual** and set a manual value for the dimensioning. For example, for a scale of **1:100mm**, you would set the manual value to **0.01**.



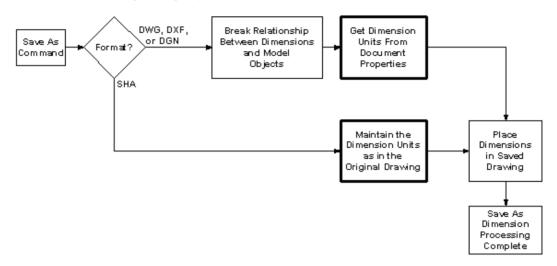
When you apply this updated *style.sha* to your drawings, you can use the new paper space dimension style when placing paper space objects, even when positioned on top of drawing view objects.



- ★ IMPORTANT Upon resizing a drawing, all automatic dimensions change accordingly, but manual dimensions will not relocate. Manual dimensions between paper space objects and model objects will still exist, but their values will change.
- **NOTE** Dimensioning from model objects in one view to model objects in a different view is not supported.

Save As MicroStation or AutoCAD Format

You can use the **Save As** command to assign dimension units and export the drawings to MicroStation or AutoCAD format. The following flowchart shows how the **Save As** command controls dimensioning during export.



Save As Command

The **Save As** command is available from the shortcut menu for any document or node containing documents in the **Console**. For orthographic drawings, the **Save As** command supports exporting to DGN, DWG, and DXF formats, as well as the native SHA formats. For more information, see *Save As Command* (on page 72) in the *Drawings Help*.

NOTE For Isogen Isometric Drawings, a file is created for each sheet in the drawing with [drawing name]_[sheet name] as the filename. For example, if the drawing My_Pipeline contains Sheet1 and Sheet2, two files will be saved with the names My_Pipeline_Sheet1 and My_Pipeline_Sheet2.

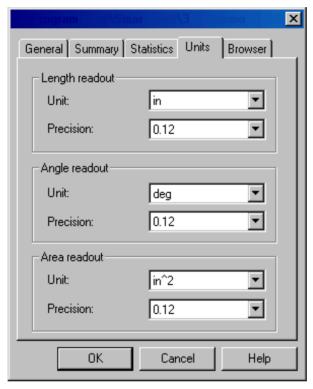
Embedded Object Dimensioning

The model graphics in the drawings are embedded in the drawing file. The dimensions in the drawings are *connected* directly to the embedded objects.

During export to MicroStation or AutoCAD formats using **Save As**, the software opens the drawings in **SmartSketch Drawing Editor** and the relationship between the embedded objects and the dimensions is broken. The dimension, however, is not removed and still maintains its original value.

Document Properties

Dimensions in drawings exported to the DGN, DWG, and DXF formats do not use the active dimension formatting for their units. Instead, the software determines dimension units from the document properties for the drawing being exported. This behavior applies whether the drawings is a volume, *by query*, or composed drawing type. You can view and edit the document units by opening the drawing in **SmartSketch Drawing Editor** and selecting **File > Properties**. You can see the current unit settings on the **Units** tab.



★ IMPORTANT We recommend that border templates have identical units and precision values in the document properties set in the active dimension style. This maintains dimension unit accuracy if you export the drawing to a foreign file format.

See Also

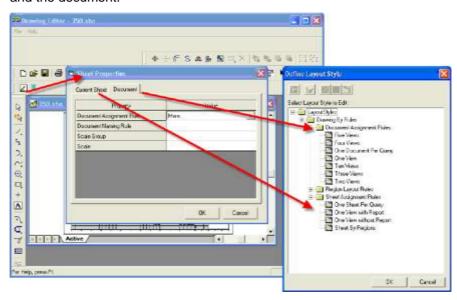
Dimensions (on page 328)

Edit Sheet Properties Command

Specifies properties for the current sheet and document. This command is available on the Drawings Compose toolbar in SmartSketch Drawing Editor when you:

- Edit a template associated with a Drawings by Rule component. Right-click the component and select **Edit Template** on the shortcut menu.
- Edit an existing drawing document. Right-click a Drawings by Rule drawing document in the
 Detail View and select Edit on the shortcut menu.

In both cases, the template or drawing document opens in SmartSketch Drawing Editor with the **Edit Sheet Properties** available on the toolbar above the drawing area. When you click the command, the **Sheet Properties** dialog box appears so you can set the properties for the sheet and the document.



See Also

Sheet Properties Dialog Box (on page 344)

Change the Border Template for an Existing Sheet

In this workflow, you change the border for an individual sheet to a template different from the border template defined in the component. For example, you may want the first sheet of drawing to use a different border template than all other sheets.

- 1. Right-click the drawing document in the Management Console Detail View, or in the Drawing Console and select **Edit**. The drawing opens in **SmartSketch Drawing Editor**.
- 2. Select a sheet tab at the bottom of the drawing view.
- 3. Click Edit Sheet Properties dialog box displays.
- 4. On the **Current Sheet** tab, select a new value for **Border Template**.
- 5. Click **OK**. The new border template is applied to the sheet.

■ NOTES

- Layout Template is inactive, and cannot be changed for an existing sheet
- Border Layout and Template Layout are customizable items.
- For more information about layout properties, see *Edit Sheet Properties Command* (on page 342) in *SmartSketch Drawing Editor Help*.

Sheet Properties Dialog Box

Specifies properties for the current sheet and document. This dialog box displays when you click **Edit Sheet Properties**, which is only available in **SmartSketch Drawing Editor** when you are editing a Drawings by Rule template or document.

See Also

Edit Sheet Properties Command (on page 342) Current Sheet Tab (on page 345) Document Tab (on page 344)

Document Tab

Specifies properties for the document. This dialog box displays when you click **Edit Sheet Properties**, which is only available in **SmartSketch Drawing Editor** when you are editing a Drawings by Ruleset template or document.

The properties available are defined as follows:

Document Assignment Rule

Specifies the layout style rule to use for the document. Select **More** in the dropdown to display the **Define Layout Style** dialog box and select the layout style to use.

Document Naming Rule

Defines how the document will be named.

Scale Group

Specifies a scale group for the document, such as Metric, Imperial, or Manual Scale. When you select a scale group, the **Scale** property values update. Selecting **Manual Scale** shows all scales, metric or imperial.

Scale

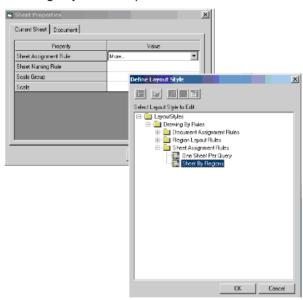
Specifies a scale to use for the document. The **Scale** values update depending on the selected **Scale Group**.

See Also

Edit Sheet Properties Command (on page 342) Sheet Properties Dialog Box (on page 344)

Current Sheet Tab

Specifies properties for the current sheet. This dialog box displays when you click **Edit Sheet Properties**, which is only available in **SmartSketch Drawing Editor** when you are editing a Drawings by Rule template or document.



The properties available are defined as follows:

Sheet Assignment Rule

Specifies the layout style rule to use for the sheet. Select **More** in the list to display the **Define Layout Style** dialog box and select the layout style to use.

Sheet Naming Rule

Defines how the sheet will be named.

Scale Group

Specifies a scale group for the sheet, such as Metric, Imperial, or Manual Scale. When you select a scale group, the **Scale** property values update. Selecting **Manual Scale** shows all scales, metric or imperial.

Scale

Specifies a scale to use for the sheet. The **Scale** values update depending on the selected **Scale Group**.

See Also

Edit Sheet Properties Command (on page 342) Sheet Properties Dialog Box (on page 344)

Drawings Compose Toolbar

This toolbar is available in SmartSketch Drawing Editor when you click **Edit** on a drawing document shortcut menu in the Drawings and Reports task. For more information, see *Drawing Document Shortcut Menu* and *Edit Command* (on page 37) in the *Drawings and Reports Reference Data Guide*.

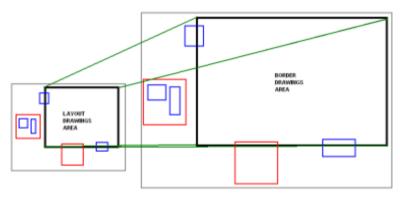
Place View Command

Creates a new drawing view in the drawing area. This command is only available in SmartSketch Drawing Editor when you create a new drawing or open an existing drawing from a 3D task.

When you click **Place View**, you click and drag to define the new view. The **Drawing View Properties** dialog box appears so that you can define the properties associated with the view. The properties shown change depending on the View Style type you select, such as Orthographic View or Key Plan View.

After creating the view and defining its properties, use the **Associate Objects to View** command to specify the content of the view. You can also move the view around, crop it, or align it in the drawing sheet.

NOTE You can create drawing views outside the drawing area. Also, when you stretch the drawing area to match the border file, the software retains your offsets so that a view or region outside the drawing area remains outside the drawing area even after it is placed in the border file.



You can create a drawing view outside a region, but this makes the drawing view an "unmanaged view," meaning the properties of the region do not impact the drawing view. However, if a view is entirely inside a region or touching a region, the region *manages* the drawing view and, when you update the drawing in a 3D task, the software pulls the drawing view into the region and updates it based on the region layout style.

See Also

Drawing View Properties Dialog Box (Place View Command) - Composed Drawings (on page 348)

Update View Command (on page 485)

Drawing View Properties Dialog Box (Place View Command) - Composed Drawings

Specifies properties for the selected drawing view in a composed drawing. You can access this dialog box when you right-click on a drawing view and then select **Properties** on the shortcut menu.

For more information on creating a composed drawing, see *Create a new composed drawing* in the *Orthographic Drawings User's Guide*.



Info Tab (Drawing View Properties Dialog Box) (on page 348)

Format Tab (Drawing View Properties Dialog Box) (on page 349)

View Tab (Drawing View Properties Dialog Box) (on page 349)

See Also

Place View Command (on page 347)

Info Tab (Drawing View Properties Dialog Box)

Specifies general view properties.

Type

Displays the category of the selected element.

Sheet

Displays the name of the drawing sheet that contains the selected element.

Laver

Shows the layer that contains the selected element.

Origin

Specifies the coordinates, or location, of an element along the X- and Y-axes.

Format Tab (Drawing View Properties Dialog Box)

Formats the frame around a drawing view.

Show Border

Displays the frame around the object.

Color

Sets the color of the frame.

Line Width

Sets the line thickness on the frame.

Line Type

Overrides a line type for a drawing sheet or embedded object and sets another line style for an element or a linked object.

View Tab (Drawing View Properties Dialog Box)

Sets the drawing view style and other properties for a selected drawing view in a composed drawing.

Style

Specifies a view style, which includes rules for filters, updates, and graphics. The view style controls the output characteristics of the view on the generated drawing. The list displays the 10 most recently used view styles in the session. Click **More...** to display the **Select View Style** dialog box.

Name

Specifies a name for the view. You must type a name in order to create a view.

View Style

Identifies the view style used within the rule set associated with this drawing document. This is a read-only field and is shown for Ruleset view styles only.

Coordinate System Properties

Annotation Coordinate System

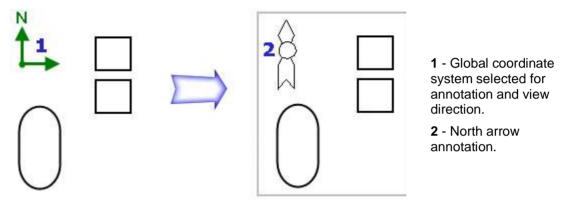
Specifies the coordinate system used to place annotations (labels and dimensions) on the drawing. This property is not supported in Ruleset view styles. Choose a coordinate system from the list, or click **More...** to choose another coordinate system with the **Select System** dialog box.

View Direction Coordinate System

Specifies the coordinate system to use with regard to the "looking direction" for the drawing view contents. The coordinates listed are defined for the model. This property is not supported in Ruleset view styles. Choose a coordinate system from the list, or click **More...** to choose another coordinate system with the **Select System** dialog box.

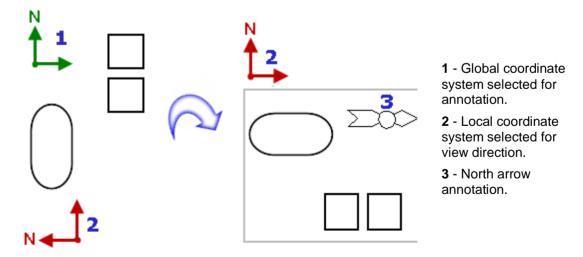
Example 1

The global coordinate system is selected for both **Annotation Coordinate System** and **View Direction Coordinate System**. A plan view is created, and the global view direction coordinate system North points up. Because the North arrow annotation is also defined by the global coordinate system, the North arrow annotation also points up.



Example 2

The global coordinate system is selected for **Annotation Coordinate System** and a local coordinate system is selected for **View Direction Coordinate System**. North on the local coordinate system is rotated 90° counter-clockwise relative to the global coordinate system. A plan view is created, for which North of the local view direction coordinate system points up, rotating the view 90° clockwise. Because the North arrow annotation is defined by the global coordinate system, the North arrow annotation points to the right.



Orientation Properties

Orientation Rule

Specifies the rules available for orientations for the current view type. The **Orientation Rule** drives the values for the **Orientation View Direction** and **Orientation Up Direction** properties. This property is only shown for Ruleset view styles.

Orientation View Direction

Indicates the elevation direction in which you want the drawing view to "look" at the drawing objects. This property is only shown for Ruleset view styles.

Orientation Up Direction

Indicates the x- and y-axis direction with which you want the drawing view to "look" at the drawing objects. This property is only shown for Ruleset view styles.

Scale Properties

Scale Family

Specifies a scale family for the drawing view, such as Metric, Imperial, or Manual Scale. When you select a scale family, the **Scale** property values update. Selecting **Manual Scale** shows all scales, metric or imperial.

User Selected Scale

Specifies a scale to use for the drawing view. The **Scale** values update depending on the selected **Scale Family**.

■ NOTES

- For Custom scale, the default is the unit of measure setting of the document. For example, if you type values of 1 in to 1 ft, the values are converted to mm if that is the default unit of measure.
- Do not use negative values when typing custom scale values.

Navigation Rule Properties

HngSupSimpleNavigator.dll

Returns support components, supporting objects, supported objects, and design children. This rule also returns the control points on the components. This rule is specific to objects created in the Hangers and Support task.

HngSupRangeNavigator.dll

Returns support components, supporting objects, supported objects, and design children collections that extend the Hangers and Support range. This rule also returns the control points on the components. This rule is specific to objects created in the Hangers and Support task, and operates similarly to the **HngSupSimpleNavigator.dll** rule.

DrawingSpoolNavigator.dll

Returns the spool, its connected parts and their features. This rule is specific to pipe spools.

AssemblyNavigator.dll

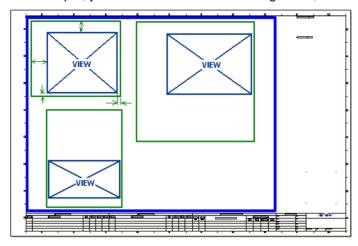
Returns the assemblies, pipe spool, penetration spool, its connected parts, and their features. This rule is specific to assemblies.

Margin Properties

Margin Left/Margin Right/Margin Top/Margin Bottom

Defines the marginal distance for the drawing view. The margin is the distance surrounding the drawing view SmartFrame. This area is used for labels and dimensions that have been designed to incorporate margins in their search for clear space.

For example, you could have several drawing views, each with different margin settings:



Other General Properties

Look Direction

Indicates the direction in which you want the drawing view to "look" at the associated volume or objects.

Description

Describes the content of the view. This description is optional.

View Offset

Expands the view on the drawing sheet in all directions in order to prevent matchline distortion and grid plane clipping. The default offset value is **2 mm** for composed views and **0 mm** for ruleset views. The offset does not affect the size of the volume that is associated with the view.

■ NOTES

- View Offset is only available for views that are associated to a volume. Section and detail views do not have this property value.
- Negative values are not permitted for this property.
- In ruleset views, if you change the view style before editing View Offset for the first time, View Offset displays the offset value as defined in the .xml template of the view style. Make sure that your View Offset value is correct before closing the Drawing View Properties dialog box.

Flush Threshold

Sets a parameter for memory management, and helps improve drawing update performance. When the number of objects processed for a drawing document reaches the **Threshold** value, they are removed from memory. If they are needed later, they are recalled from the database. The **Flush Threshold** property is only available for composed drawing documents. The default value is **2000**, with a range of **5-5000**. Higher values are faster but use more memory, which is acceptable for smaller drawings. Lower values are slower but allow larger drawings to complete faster.

NOTE If a drawing document does not successfully update in the Drawings and Reports 3D task, check the error log for the drawing document. If the error log shows memory overflow errors, lower the **Flush Threshold** value.

VHL Precision

Sets a parameter for Hidden Line removal processing to consider two lines as identical. This property setting has a direct impact on the visibility of the lines in the drawing. It improves drawing update performance, but it may reduce drawing quality. The **VHL precision** property is only available for composed drawing documents. The default value is **0.000001**, with a range of **0.001 to 0.000001**. The smaller the value, the more accurate the graphic elements are in the 2D view.

NOTE If some intersections of complex surfaces appear on/off along the curve, the precision of the VHL may be too restrictive compared with the precision by which the surfaces were created. Lowering the **VHL Precision** value may help the display of the intersections, but lowering the value too much could degrade the overall quality and the visibility of the drawing details.

Geometry Validation

Sets a parameter for drawing completion and quality to improve drawing update performance. The **Geometry Validation** property is available for composed drawing documents only. The default value is **Off**. When set to **Off**, the drawing document completes, but invalid geometries are left out. If set to **On**, the drawing document does not complete if invalid geometries are encountered during update.

Angle for Target Evaluation (Marine mode only)

Defines the allowable angle from the view plane for a target plate. The **Angle for Target Evaluation** property is available for the scantling view style only. Additionally, this property is used only with the **Place Detail View** or the **Place Section View** command in the SmartSketch Drawing Editor and only when a reference plane selected for the **Any system, part, or reference plane** Drawing by Rule query has been set. When plates or profiles are selected in the query, target evaluation is determined by a combination of levels and type of connection. The default value is 20°.

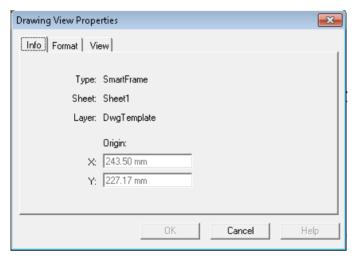
■ NOTES

- The angle is displayed in the units specified in the session file, which is degrees by default.
- For more information about **Any system, part, or reference plane**, see *Drawings by Rule Queries* in Queries Tab (Setup Dialog Box Drawings by Rule).

Drawing View Properties Dialog Box (Place View Command) - Steel Order Drawings

Specifies properties for the selected drawing view in a steel order drawing. You can access this dialog box when you select and right-click on a drawing view and then select **Properties** on the shortcut menu.

For more information on creating a scantling drawing, see the *Orthographic Drawings User's Guide*.



Info Tab (Drawing View Properties Dialog Box) (on page 348)

Format Tab (Drawing View Properties Dialog Box) (on page 349)

View Tab (Drawing View Properties Dialog Box - Drawings View Explorer) (on page 355)

See Also

Place View Command (on page 347)

Info Tab (Drawing View Properties Dialog Box)

Specifies general view properties.

Type

Displays the category of the selected element.

Sheet

Displays the name of the drawing sheet that contains the selected element.

Laver

Shows the layer that contains the selected element.

Origin

Specifies the coordinates, or location, of an element along the X- and Y-axes.

Format Tab (Drawing View Properties Dialog Box)

Formats the frame around a drawing view.

Show Border

Displays the frame around the object.

Color

Sets the color of the frame.

Line Width

Sets the line thickness on the frame.

Line Type

Overrides a line type for a drawing sheet or embedded object and sets another line style for an element or a linked object.

View Tab (Drawing View Properties Dialog Box - Drawings View Explorer)

Sets the drawing view style and other properties for a selected drawing view in a steel order drawing.

Style

Specifies a view style, which includes rules for filters, updates, and graphics. The view style controls the output characteristics of the view on the generated drawing. The list displays the 10 most recently used view styles in the session. Click **More...** to display the **Select View Style** dialog box.

NOTE (Marine mode only) To display only RulesetStyles in marine mode, you need to remove the marine and material handling productIDs from the KeyPlan.xml, Orthographic.xml, and Spatial.xml files as shown in the example below. The .xml files are located in the [Reference Data Folder]\SharedContent\Drawings\Catalog\MetaStyles folder.

Original

Name

Specifies a name for the drawing view.

Coordinate System Properties

Coordinate System

Specifies a coordinate system to include in the drawing.

Orientation Properties

Orientation Rule

Specifies the rules available for orientations for the current view type. The **Orientation Rule** drives the values for the **Orientation View Direction** and **Orientation Up Direction** properties. This property is only shown for rule set view styles.

Orientation View Direction

Indicates the elevation direction in which you want the drawing view to "look" at the drawing objects. This property is only shown for rule set view styles.

Orientation Up Direction

Indicates the x- and y-axis direction with which you want the drawing view to "look" at the drawing objects. This property is only shown for rule set view styles.

Scale Properties

Scale Family

Specifies a scale family for the drawing view, such as Metric, Imperial, or Manual Scale. When you select a scale family, the **Scale** property values update. Selecting **Manual Scale** shows all scales, metric or imperial.

User Selected Scale

Specifies a scale to use for the drawing view. The **Scale** values update depending on the selected **Scale Family**.

■ NOTES

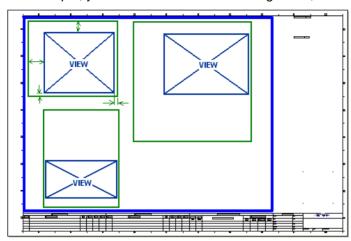
- For Custom scale, the default is the document's unit of measure setting. For example, if you type values of 1 in to 1 ft, the values are converted to mm if that is the default unit of measure.
- Do not use negative values when typing custom scale values.

Margin Properties

Margin Left/Margin Right/Margin Top/Margin Bottom

Defines the marginal distance for the drawing view. The margin is the distance surrounding the drawing view SmartFrame. This area is used for labels and dimensions that have been designed to incorporate margins in their search for clear space.

For example, you could have several drawing views, each with different margin settings:



Other General Properties

Description

Provides a description of the drawing view. This description is optional.

View Offset

Expands the view on the drawing sheet in all directions in order to prevent matchline distortion and grid plane clipping. The default offset value is **2 mm** for composed views and **0 mm** for ruleset views. The offset does not affect the size of the volume that is associated with the view.

■ NOTES

- View Offset is only available for views that are associated to a volume. Section and detail views do not have this property value.
- Negative values are not permitted for this property.
- In rule set views, if you change the view style before editing View Offset for the first time, View Offset displays the offset value as defined in the .xml template of the view style. Make sure that your View Offset value is correct before closing the Drawing View Properties dialog box.
- View annotations, such as the ruler and view name, are automatically adjusted in relation to the View Offset value.

VHL Precision

Sets a parameter for Hidden Line removal processing to consider two lines as identical. This property setting has a direct impact on the visibility of the lines in the drawing. It improves drawing update performance, but it may reduce drawing quality. The **VHL precision** property is only available for composed drawing documents. The default value is **0.000001**, with a range of **0.001 to 0.000001**. The smaller the number, the more accurate the graphic elements are in the 2D view.

Geometry Validation

Sets a parameter for drawing completion and quality to improve drawing update performance. The **Geometry Validation** property is available for composed drawing documents only. The default value is **Off**. When set to **Off**, the drawing document completes, but invalid geometries are left out. If set to **On**, the drawing document does not complete if invalid geometries are encountered during update.

View Cone Angle

Displays the cone angle value. This property is only available when you select the **Steel Order Extensions.xml** file as the **Additional View Inputs** property value in the **Edit Ruleset View Style** dialog box. For more information, see *General Tab (Edit Ruleset View Style Dialog Box)* (Edit Ruleset View Style Dialog Box) in the Smart 3D Drawings and Reports Reference Data Guide.

See Also

Drawings View Explorer (on page 324)

Automatic Resize Behavior of Composed Views

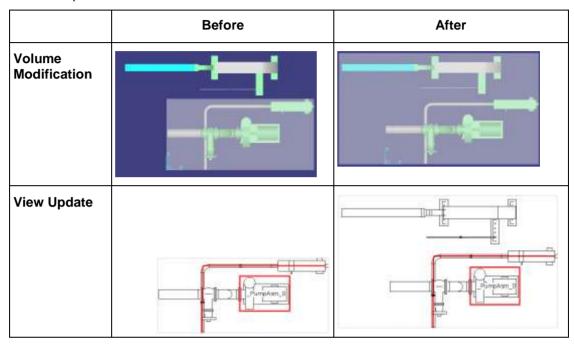
For Composed drawings, views that are too small to display the associated volume or are larger than the volume are automatically resized to fit unless the view is set to **Fit to Scale** or is managed by a region. The view size grows or shrinks from the center of the view and view proportions may change after the resize. The drawing must be saved to make the resize of the view permanent.

NOTE You cannot crop a composed drawing view that is scaled. Views that are set to **Fit to Scale** can be cropped.

The following examples are common workflows that are affected by the automatic resizing behavior of the composed drawing view.

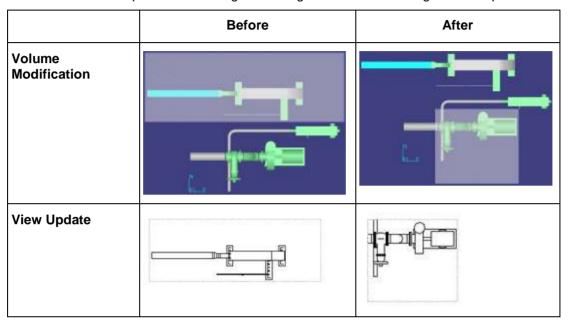
Overlapping Volumes Modification

If you manually shift the sides of an associated volume such that the new volume location overlaps a portion of the old volume, the associated view resizes to mirror the volume changes. You must update the drawing or drawing view for these changes to take place. Manually-placed graphics and labels stay in the correct location on the drawing after the volume size modification and view update.



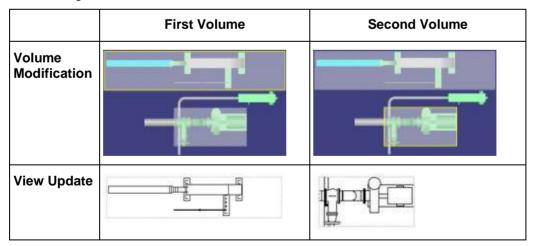
Non-overlapping Volumes Modification

If you manually shift the sides of an associated volume, or move the volume, such that the new volume location does not overlap a portion of the old volume, the associated view resizes symmetrically around the previous center point of the view. If the volume is moved to a new location, the view does not resize. In either case, the view contents are updated with the volume contents. You must update the drawing or drawing view for these changes to take place.



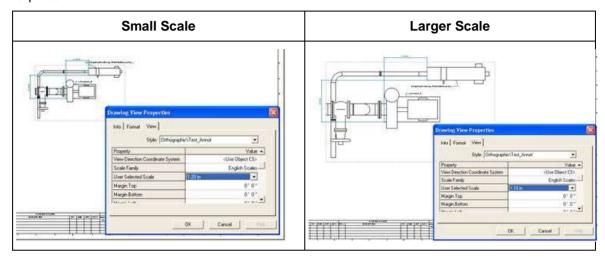
View to Volume Association

If you associate a view to a volume with the **Associate Objects to View** command, the view resizes symmetrically around the center point to match the volume size. The view resizes to match a larger or smaller volume.



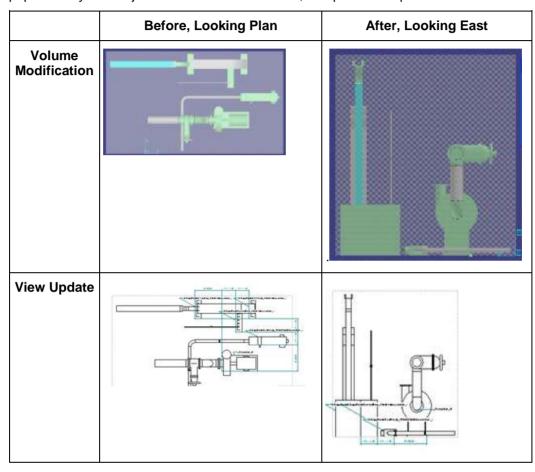
View Scale Modification

If you change the view scale on the **Drawing View Properties** dialog box, the view resizes symmetrically around the center point to accommodate the new scale. Depending on the scale change, the view grows or shrinks in size. You do not need to update the view to see these changes, but in order to populate any new objects that are inside the view, an update is required.



View Look Direction Modification

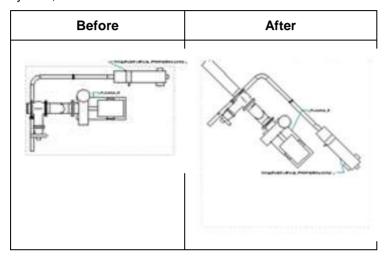
If you change the **Look Direction** from the **Drawing View Properties** dialog box, the view resizes symmetrically around the center point to match the volume size according to the new look direction. You do not need to update the view to see these changes, but in order to populate any new objects that are inside the view, an update is required.



View Direction Coordinate System Modification

If you change the **View Direction Coordinate System** in the **Drawing View Properties** dialog box, the view resizes symmetrically around the center point to accommodate the new view direction coordinate system. You do not need to update the view to see these changes, but in order to populate any new objects that are inside the view, an update is required.

NOTE If the new coordinate system has the same orientation as the previous coordinate system, the view does not resize.



Place a Manual View

The following workflows allow you to place a manual view in a Ruleset drawing either by Parts or by Reference Plane and Block.

See Also

Place a Manual View (By Parts) (on page 363)
Place a Manual View (By Reference Plane and Block) (on page 364)

Place a Manual View (By Parts)

In this workflow, you directly select detailed parts to associate with the view.

- 1. In Smart 3D, switch to a task other than Drawings and Reports, such as Molded Forms or Structural Detailing.
- 2. Select Tools > Drawing Console.

The **Drawing Console** dialog box displays.

3. Right click a drawing document and select Edit.

SmartSketch Drawing Editor displays. In the Drawings View Explorer, a single sheet displays as a child of the drawing document.

4. In SmartSketch Drawing Editor, click the **Place View** command. Click and drag in the drawing area to place a graphic view.

The **Drawing View Properties** dialog box displays.

5. On the View tab, select More for the Style property.

The Select View Style dialog box displays.

6. Navigate through the hierarchy to **Ruleset Styles**. Select an appropriate steel order ruleset view style, such as **Steel Order (Decks)** or **Steel Order (Generic)**, and click **OK**.

Additional properties display in the View tab of the Drawing View Properties dialog box.

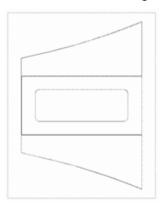
- 7. On the View tab, type a value for Name.
- 8. On the View tab, select the needed values for Coordinate System, Scale Family, and User Selected Scale.
- 9. On the View tab, select Scantlings for Plate / Profile as the Orientation Rule value.
- 10. Click **OK**.

In the Drawings View Explorer, the view displays as a child of the drawing sheet.

- 11. Select the graphic view in the drawing area and click Associate Objects to Views 38.
- 12. Select the 3D application window to make it the active window.

The Associate ribbon displays in the 3D window.

- 13. To associate parts, select **1. Structural Parts or Plane** as the value for **Query** on the **Associate** ribbon.
- 14. In the Workspace Explorer, select plates to associate to the view. Select detailed parts, light (non-detailed) parts, systems, or leaf systems.
- 15. Click **Finish** on the **Associate** ribbon to complete the association to the drawing view. In SmartSketch Drawing Editor, preview geometry of the selected plate parts displays.



Place a Manual View (By Reference Plane and Block)

In this workflow, you select a grid plane or an offset from a grid plane to associate parts with the view and select a block, assembly block or assembly to clip the plane selected. You can use the workflow to place a major view, such as the view for a deck, transverse bulkhead, or longitudinal bulkhead.

- 1. In Smart 3D, switch to a task other than Drawings and Reports, such as Molded Forms or Structural Detailing.
- 2. Select Tools > Drawing Console.

The **Drawing Console** dialog box displays.

3. Right click a drawing document and select Edit.

SmartSketch Drawing Editor displays. In the Drawings View Explorer, a single sheet displays as a child of the drawing document.

4. In SmartSketch Drawing Editor, click the **Place View see** command. Click and drag in the drawing area to place a graphic view.

The **Drawing View Properties** dialog box displays.

5. On the View tab, select More for the Style property.

The **Select View Style** dialog box displays.

6. Navigate through the hierarchy to **Ruleset Styles**. Select an appropriate steel order ruleset view style, such as Steel Order (Reference Plane) or Steel Order - Reference Plane (Generic), and click OK.

Additional properties display in the View tab of the Drawing View Properties dialog box.

- 7. On the **View** tab, type a value for **Name**.
- 8. On the View tab, select the needed values for Coordinate System, Scale Family, and User Selected Scale.
- 9. On the View tab, select Scantlings for Plate / Profile as the Orientation Rule value.
- 10. Click **OK**.

In the Drawings View Explorer, the view displays as a child of the drawing sheet.



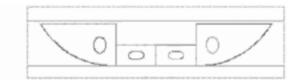
- 11. Select the graphic view in the drawing area and click **Associate Objects to Views** 38.
- 12. Select the 3D application window to make it the active window.

The Associate ribbon displays in the 3D window.

- 13. To associate parts by grid plane, select 1. Structural Parts or Plane as the value for Query on the Associate ribbon. In the Workspace Explorer, select a reference plane coincident with the needed plate parts. For the example in this workflow, a transverse reference plane is selected.
- 14. To associate parts by an offset from a grid plane, select 7. Offset (Optional) as the value for **Query** on the **Associate** ribbon. In the Workspace Explorer, select a reference plane. Type an offset value in **Value** on the **Associate** ribbon.
- 15. Select plate parts to associate to the view. For the example in this workflow, select plate parts.

- Select 2. Block, Volume or Assembly (Optional) as the value for Query on the Associate ribbon.
- 17. In the Workspace Explorer, select the **Assembly** tab.
- 18. Select a block, assembly block or assembly that will define the extents of the plane that is associated with the view being created.
- 19. Click Finish on the Associate ribbon to complete the association to the drawing view.

In SmartSketch Drawing Editor, preview geometry of the plate parts displays. Only parts within the selected block or assembly and coincident with the selected reference plane are associated with the view. The block boundaries are also displayed.



Place a Manual View for Non-Shell Plates

In this workflow, you use additional options in the **Associate** ribbon to place a manual view for non-shell plates. These options display when you select the **Use Expansion** query in the **Query** tab in the **Edit Ruleset View Style** dialog box. For more information, see Edit Ruleset View Style Dialog Box.

- In Smart 3D, switch to a task other than Drawings and Reports, such as Molded Forms or Structural Detailing.
- 2. Select Tools > Drawing Console.

The **Drawing Console** dialog box displays.

3. Right click a drawing document, and select Edit.

SmartSketch Drawing Editor displays. In the Drawings View Explorer, a single sheet displays as a child of the drawing document.

4. In SmartSketch Drawing Editor, click the **Place View** command. Click and drag in the drawing area to place a graphic view.

The **Drawing View Properties** dialog box displays.

5. On the View tab, select More for the Style property.

The **Select View Style** dialog box displays.

6. Navigate through the hierarchy to **Ruleset Styles**. Select an appropriate steel order ruleset view style, such as **Steel Order (Reference Plane)**, and click **OK**.

Additional properties display in the View tab of the Drawing View Properties dialog box.

- 7. On the **View** tab, type a value for **Name**.
- 8. On the View tab, select the needed values for Coordinate System, Scale Family, and User Selected Scale.

- 9. On the View tab, select Shell Expansion for Plate as the Orientation Rule value.
- 10. Click **OK**.

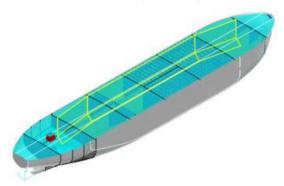
In the Drawings View Explorer, the view displays as a child of the drawing sheet.

- 11. Select the graphic view in the drawing area, and click Associate Objects to Views 38.
- 12. Select the 3D application window to make it the active window.

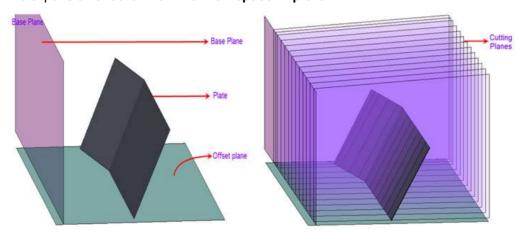
The Associate ribbon displays in the 3D window.

13. Click **Expansion Surface**, and then select the root or leaf plate systems to see in the manual view.

NOTE You can only select systems, not parts.



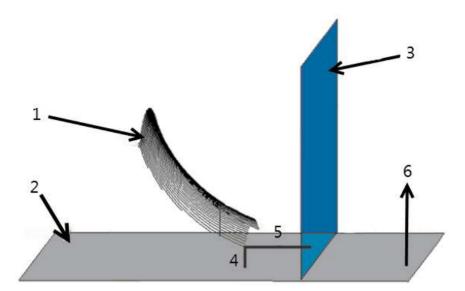
14. Click **Cutting Plane Axis** , and then select the axis whose child planes are parallel to the expansion direction from the **Workspace Explorer**.



NOTE Girth lengths are measured along curves defined by the intersection of each plate system and each cutting plane.

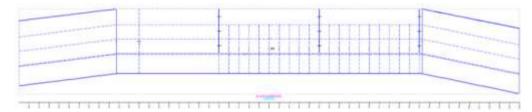
15. Click **Base Plane Definition** , and then select the base plane that defines the start point for girth measurements.

16. Specify any additional options as necessary. For example, you can select the offset plane, as shown below. For more information, see *Associate Objects to View Command* (on page 374).



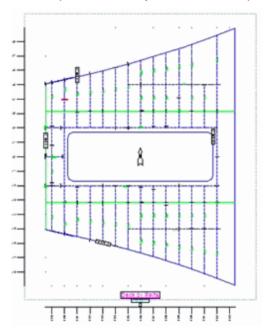
- 1 Plate Curve
- 2 Offset Plane
- 3 Base Plane
- 4 Distance from the offset plane to the start point of the plate curve
- 5 Distance from the base plane to the start point of the plate curve
- 6 Normal of the offset plane pointing to the plate
- 17. Click **Finish** on the **Associate** ribbon to complete the association to the drawing view.

 In SmartSketch Drawing Editor, preview geometry of the selected non-shell plates displays.



Update a Manual View

- 1. Right-click the out-of-date view in the Drawings View Explorer and select **Update**.
- 2. After the view updates, the view frame in the graphics area shows all geometry defined by the component view style. The view is up-to-date in the Drawing View Explorer.



■ NOTES

- You can delete a manual view permanently, or select UnAssign to move the view in the UnAssigned Folder. If you delete a view that is a parent of other views, such as a detail or section view, the Convert or Delete dialog box displays. Select Convert to independent drawing view(s) to save the child view as an independent drawing view, or select Delete to delete the child view.
- A sheet cannot be deleted unless all views on the sheet are removed first.
- All views within the drawing must be up-to-date or the drawing status will be out-of-date.
- For information on the 2D commands available for editing, see the *SmartSketch Drawing Editor Help*.

Move a Drawing View

Plant, marine, and material handling mode drawing views can only be moved while holding the **ALT** key when selecting the view. This includes drawing, section, key plan, and detail views that contain any graphics.

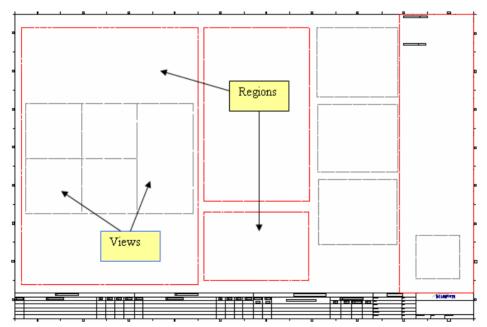
- 1. Press and hold the ALT key.
- 2. Select a view.
- 3. Drag the view to a new location.

■ NOTES

- Drawing views that have not been updated can be moved without pressing the ALT key.
- Empty views and embedded reports views can be moved without pressing the ALT key.
- The Move, Nudge, Drag, and Select All commands found on the Change toolbar also requires the use of the ALT key when selecting a view.
- You must use the Fence Select or Select All commands to select multiple views. You cannot use the CTRL key to select multiple views.

Place Region Command

Creates a new drawing region in the drawing area. A *region* is a container that controls the arrangement of the views inside by means of an associated layout style. Views controlled by a region are called *managed* views. You can manage any type of drawing view (report, key plan and graphic views) with a region.



This command is available in **SmartSketch Drawing Editor** when you edit a layout template or edit an existing drawing.

When you click **Place Region**, you click and drag to define the new region. The **Region Properties** dialog box appears so that you can define the properties associated with the region.

After creating the region and defining its properties, use the **Place View** or **Place Report View** command to place views within the region.

Region Behavior

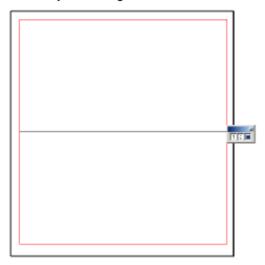
- You can place a drawing view outside a region, but this makes the drawing view an "unmanaged view," meaning the properties of the region do not control the drawing view. However, if a view is inside a region or touching a region, the region manages the drawing view and, when you update the drawing in the 3D Drawings and Reports task, the software pulls the drawing view into the region and updates it based on the region layout style and layout processor.
- If a region refuses to accept a drawing view (for example: the region is defined for three views, and you are attempting to add a fourth view), the drawing view is added as an unmanaged view, just outside and to the upper left of the region. If another region occupies this space in the drawing area, the unmanaged view is placed as close to the original region as possible.
- If a drawing view "straddles" two or more regions, the region that contains more of the drawing view manages it. If the drawing view equally straddles two or more regions, the software uses the first drawing view point to measure distance and determine which region manages the drawing view.
- If a region contains a drawing view and the drawing view properties make it ineligible for the region, the software removes the drawing view from the region automatically and places it in the upper left of the drawing area, outside of other regions.
- If you switch the border of a drawing using the Switch Border command, views may reposition or resize with the new border template based on the following conditions:
 - Managed views are resized and repositioned according to the new border template.
 - Unmanaged views are not resized and repositioned.

For more information, see Switch Border (on page 485).

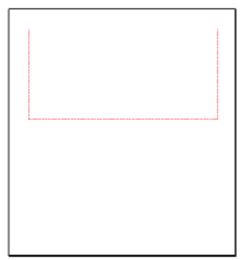
Deleting a Region

Regions are used to manage the positioning of views within a layout template. When you delete a region, any views that were managed by the region become "unmanaged" views.

1. To delete a region from a layout template, you should use **PickQuick** to select the outer boundary of the region.



2. After selecting the region outer boundary, select **Edit > Delete** to remove the region definition. You can also press the **Delete** key on the keyboard. Any views placed within the region remain on the layout template as unmanaged views.



See Also

Region Properties Dialog Box (on page 373)

Region Properties Dialog Box

Specifies properties for the selected region. This dialog box displays after you have created a new region using the **Place Region** command, which is only available in **SmartSketch Drawing Editor**.

Description

Provides a description of the region.

Layout Style

Specifies the layout style to associate with this region. The layout style dictates the order and placement of the drawing views managed by the region. Select **More** to display the **Define Layout Style** dialog box and select the layout style from a list of available styles. For more information, see *Define Layout Style Dialog Box* (on page 373).

Maintain Aspect

Specifies whether or not the aspect ratio for each drawing view and region is maintained within the layout. This property is useful when you are "stretching" a layout into a border or when you are changing the border.

Lock Region

Locks the positioning of the drawing views within the region. This property also prevents you from adding or deleting existing views.

See Also

Convert Excel Spreadsheet Reports to Native Text Box Format Custom Command (on page 491)

Place Region Command (on page 370)

Define Layout Style Dialog Box

Specifies a layout style to associate with a drawing region. This dialog box displays when you select **More** in the **Value** field for the **Layout Style** property on the **Region Properties** dialog box. The hierarchy shows all available layout styles.

See Also

Convert Excel Spreadsheet Reports to Native Text Box Format Custom Command (on page 491)

Place Region Command (on page 370)

Region Properties Dialog Box (on page 373)

Place Snapshot View Command

Creates a new snapshot drawing view in the drawing area. This command is only available in **SmartSketch Drawing Editor** when you create a new drawing or open an existing drawing from a 3D task.

NOTE Before you place snapshot views, you need to create a snapshot using the 3D task **Tools > Snapshot View** command. If you do not have any snapshot views, a message displays and you can go back to a 3D task and create the snapshot views.

When you click **Place Snapshot View** , you click and drag to define the new view. You specify the snapshot view properties in the **Snapshot View** ribbon displayed above the drawing area.



When you edit a drawing that contains a snapshot view, you can right-click the view and select **Properties** to change the style or scale used in the snapshot view.

To update the contents of the snapshot view, right-click the view and select **Update View**.

See Also

Update View Command (on page 485)
Place Snapshot View Ribbon (on page 374)
Place Snapshot View Command (on page 374)

Associate Objects to View Command

Allows you to interactively specify volumes or other model objects as input for the selected drawing views. If the view is a key plan or report view, you specify an existing drawing view. This command is also used in creating composed drawings in a 3D task. The command is only available in **SmartSketch Drawing Editor** when you create a new drawing or open an existing drawing from a 3D task.

When you select a view and click **Associate Objects to View** turns off **2D/3D**, the **Associate Objects to View** turns off **2D/3D**

Selection so that you can move back and forth between the 3D application and SmartSketch Drawing Editor, selecting views and associating volumes or model objects.

Orthographic View Ribbon

If you are associating volumes to a graphic drawing view, the following ribbon options appear:

Select View

Shows either all of the graphic views in the drawing or a list of selected views from the drawing. You can multi-select views before running the **Associate Objects to View** command to populate the list with only the selected views. As you click volumes to associate them to the currently shown view, the list scrolls to the next view automatically. The views are listed in alphabetical order.



Exclude Objects

Allows you to select objects to exclude from the drawing view or to view objects that have been previously selected for exclusion from the drawing view. When you click Exclude Objects, the Accept/Cancel buttons enable on the ribbon so that you can confirm the selection. Excluded objects are highlighted.

■ NOTES

- You should associate your volume before excluding objects from the volume.
- You can select the objects from the model, or from the Workspace Explorer, but you cannot select referenced objects that originate from files on the Reference tab of the Workspace Explorer.
- You can clear objects that have previously been excluded from the drawing view by clicking on the object.
- If you exclude an object while the related tab of the Workspace Explorer is active, the child objects are automatically excluded. For example, if the System tab is active when you exclude a system object, then all of the children objects are also excluded. If the related WSE tab is not active, only the object that you specifically select is excluded. For example, if you exclude a pipe run while the Space tab is active, the child parts of the pipe run are not excluded.



Select Volume

Sets the command in select mode so that you can select a volume graphically or from the Workspace Explorer. After you select a volume in the workspace window, you can use the other controls on the ribbon to further define the objects collected for the drawing view. This is the default setting when the command ribbon appears.

X Cancel

Clears all objects selected for exclusion from the drawing view and returns you to the previous Select volume command step.

✓ Accept

Accepts all objects selected for exclusion from the drawing view and returns you to the Select volume command step.

Filter

Specifies a filter to restrict the objects to include in the volume. You can right-click and select More to select a filter from an existing list of filters or to create a new filter.

■ NOTES

- You should not use filters from the My Filters folder when using the Filter option on the Associate Objects to View ribbon.
- Section and detail views inherit filters used by the Filter option from the parent view.

Properties

Displays the properties for the selected filter.



Clear Filter

Clears the currently selected filter. This button only enables when you have selected a filter in the **Filter** dropdown field.

Look Direction

Indicates the view direction for the drawing view. The current setting is taken from the drawing view properties, set when the view was created with the **Place View** command in SmartSketch Drawing Editor. If you change the **Look Direction**, you are also changing the view direction property on the drawing view. If you select **From Active View**, the software uses the **Look Direction** of the active graphic view.

Rule Set View Ribbon (List-Based)

When the graphic view has a rule set view style, the following ribbon options appear:

Query

Specifies the query to apply to the drawing view. The selected query tells the software which objects to include in the drawing view.

NOTE A view query specifies the type of model object or parameter value used by the view definition. The list of available queries is specific to the selected package. If you are a reference data administrator customizing a drawing package, view queries are parts of the rule set defined on the **Actions** tab of the **Edit Ruleset View Style** dialog box in the Drawings and Reports task. For more information, see the appropriate drawing type under *Drawings by Rule View Style Rule Sets* in the *Drawings and Reports Reference Data Guide*.

Value

Specifies a value required by the selected query. The type of value is defined by the query. You may need to select a value from a list, type a value, or turn an option on or off.

Finish

Saves the specified query and value to the selected drawing view.

Rule Set View Ribbon (Step-Based)

When the graphic view has a rule set view style, the following ribbon options appear:

Associate Objects to View

Displays the default parts available for the view. Select the needed parts to add them to the view.

- If all needed parts are displayed, select them in the graphic view, and then click Finish to return to SmartSketch Drawing Editor.
- If all needed parts are not displayed, select them using the **Create Volume**

 or

 Gather Objects

 options.

Changes the default volume definition for the drawing view. For more information, see *Create Volume Options*, below.

Gather Objects

Changes the default part type definitions to include in the volume. For more information, see *Gather Objects Options*, below.

Orient View

Changes the default view direction and orientation. For more information, see *Orient View Options*, below.

TIP With this option, you can orient the drawing view by the local coordinate system of a part selected in the graphic view.

Finish

Saves the specified options to the drawing view and returns to SmartSketch Drawing Editor.

Reject

Rejects the selected options.

Accept

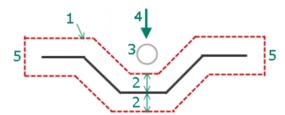
Accepts the selected options.

Create Volume Options

Volume Creation Rule - Defines the volume for the view by one of the following methods:

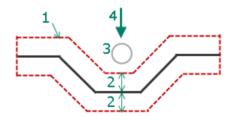
Create volume along surface of input parts with boundary extents - The volume follows the surfaces of the selected parts resulting from Any system, part, or reference plane. The volume depth (normal to the view) is extended by the values of Volume growth into drawing and Volume growth out of drawing. The volume area (in the plane of the view) is extended to (or clipped by) the boundaries defined by Boundary (Block, volume, or assembly).

Example: Corrugated bulkhead



- 1 Cross-section of volume
- 2 Volume growth
- 3 Additional part not gathered into view
- 4 View direction
- 5 Volume extended to boundaries
- Create volume along surface of input parts without boundary extents The volume follows the surfaces of the selected parts resulting from Any system, part, or reference plane. The volume depth (normal to the view) is extended by the values of Volume growth into drawing and Volume growth out of drawing. The volume area (in the plane of the view) is not extended to (nor clipped by) the boundaries defined by Boundary (Block, volume, or assembly).

Example: Corrugated bulkhead

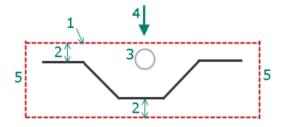


- 1 Cross-section of volume
- 2 Volume growth
- 3 Additional part not gathered into view
- 4 View direction

.

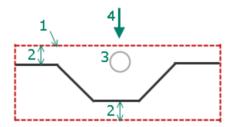
Create volume from range box around input parts with boundary extents - The volume is a rectangular range box around the selected parts resulting from Any system, part, or reference plane. The volume depth (normal to the view) is extended by the values of Volume growth into drawing and Volume growth out of drawing. The volume is extended to (or clipped by) the boundaries defined by Boundary (Block, volume, or assembly). See the corrugated bulkhead example below.

Example: Corrugated bulkhead

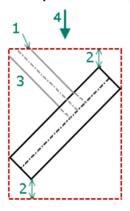


- 1 Cross-section of volume
- 2 Volume growth
- 3 Additional part gathered into view
- 4 View direction
- **5** Volume extended to boundaries
- Create volume from range box around input parts without boundary extents The volume is a rectangular range box around the selected parts resulting from Any system, part, or reference plane. The volume depth (normal to the view) is extended by the values of Volume growth into drawing and Volume growth out of drawing. The volume is not extended to (nor clipped by) the boundaries defined by Boundary (Block, volume, or assembly). See the corrugated bulkhead example below.

Example: Corrugated bulkhead



Example: Non-orthogonal member



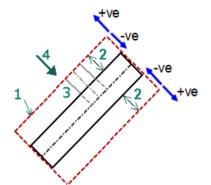
- 1 Cross-section of volume
- 2 Volume growth
- 3 Additional part gathered into view
- 4 View direction

Create oriented volume around input parts without boundary extents - The volume is a rectangular box around the parts. The box is oriented around parts to minimize the volume. The parts are typically not orthogonal, but the view direction is typically normal to the box. The volume depth (normal to the view) is extended by the values of Volume growth into drawing and Volume growth out of drawing.

■ NOTES

- If the view direction is not normal to the box, then the volume is extended to (or clipped by) the boundaries defined by Boundary (Block, volume, or assembly) instead of Volume growth into drawing and Volume growth out of drawing.
- If the view direction is normal to the box, then the volume is extended to (or clipped by) Volume growth into drawing and Volume growth out of drawing. These values can be positive or negative.

Example: Non-orthogonal member



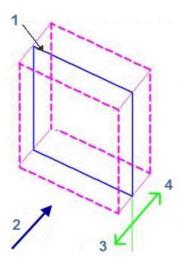
- 1 Cross-section of volume
- 2 Volume growth
- 3 Additional part gathered into view
- 4 View direction
- **Do not create volume** A volume is not used to restrict gathering using gathering rule. For example, a shell longitudinal profile view does not use a volume. It instead uses levels to gather secondary parts.

Depth Into

Defines a distance into the drawing to extend the depth of a thin volume. Parts within the thin volume are displayed in the view. A value for this query is optional.

Depth Out Of

Defines a distance out of the drawing to extend the depth of a thin volume. Parts within the thin volume are displayed in the view. A value for this query is optional.



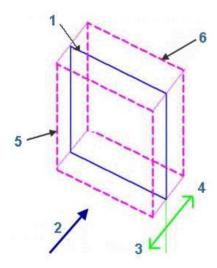
- 1 View plane
- 2 View direction
- 3 Out of drawing
- 4 Into drawing

Boundary Overlap

Defines the distance that parts extend past the view boundary.

Offset

Defines the distance from a reference plane used to create the view plane. A positive value offsets the view plane out of the drawing. A negative value offsets the view plane into the drawing. This value can be used in combination with **Volume growth into drawing** and **Volume growth out of drawing** to create a thin volume. A value for this query is optional.



- 1 Reference plane
- 2 View direction
- 3 Offset out of drawing (positive value)
- **4** Offset into drawing (negative value)
- 5 View plane offset out of drawing
- 6 View plane offset into drawing

Clip to Volume

Clips part visibility in the graphic view to the boundaries of the volume.

Gather Objects Options

Gathering Rule

Defines the type of parts gathered in the drawing view. The rule provides a definition of the objects to be gathered and drawn in a view. Select a rule that supports the type of drawing view you want to create, such as **Steel Order**, **Piping**, **Pipe Supports**, or **Assembly**.

Include all objects in volume

Defines how gathered parts are included in the drawing view. Select to gather all parts in the volume even if they are not connected to the selected parts. Clear to gather only the selected parts and parts connected to the selected parts.

📴 Show Objects Gathered

Highlights the gathered parts.

Orient View Options

Orientation Rule

Specifies the rule used to define the view orientation. Select options such as Global Coordinate System, Local Coordinate System, Scantlings for Plate/Profile, and Scantling By Plate Normal.

View Direction

Defines the direction of the view. The available options are defined by the selected orientation rule.

Up Direction

Defines the up direction of the view. The available options are defined by the selected orientation rule.

Orient Display to View Direction

Changes the orientation of the graphic view to match the view orientation. This option is only available when **Local Coordinate System** is selected for **Orientation Rule**.

For more information on view orientation options, see *Orientation Rules* in the *Smart 3D Orthographic Drawings User's Guide*.

Key Plan or Report View Ribbon

When the view you selected is specified as a report or key plan view style, you need to associate a drawing view to populate the contents of the report or key plan. The following ribbon options appear:

View

Displays a list of views available for association with the report or key plan.

Finish

Returns you to SmartSketch Drawing Editor with the selected view associated to the report or key plan view.

Generic View Ribbon

When the selected graphic view is specified as a generic rule set view style, the following ribbon options display:

Associate Objects to View

Displays the default parts available for the view. Select the needed parts to add them to the view.

- If all needed parts are displayed, select them in the graphic view, and then click **Finish** to return to SmartSketch Drawing Editor.
- If all needed parts are not displayed, select them using the **Create Volume** or **Gather Objects** options.

Changes the default volume definition for the drawing view. For more information, see *Create Volume Options*, below.

Gather Objects

Changes the default part type definitions to include in the volume. For more information, see *Gather Objects Options*, below.

Orient View

Changes the default view direction and orientation. For more information, see *Orient View Options*, below.

TIP With this option, you can orient the drawing view by the local coordinate system of a part selected in the graphic view.

Finish

Saves the specified options to the drawing view and returns to SmartSketch Drawing Editor.

Reject

Rejects the selected options.

Accept

Accepts the selected options.

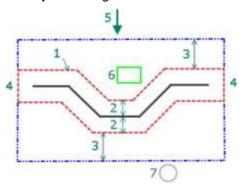
Create Volume Options

Volume Creation Rule

Defines the volume for the view by one of the following methods:

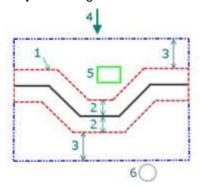
Create volume along surface of input parts with boundary extents - The volume follows the surfaces of the selected parts resulting from Any system, part, or reference plane. The volume depth (normal to the view) is extended by the values of Volume growth into drawing and Volume growth out of drawing. The volume area (in the plane of the view) is extended to (or clipped by) the boundaries defined by Boundary (Block, volume, or assembly). The gathering range is extended by the values defined by Extend Into and Extend Out Of.

Example: Corrugated bulkhead



- 1 Cross-section of volume
- 2 Volume growth
- 3 Extended gathering range
- 4 Volume extended to boundaries
- 5 View direction
- 6 Additional part gathered into view
- 7 Additional part not gathered into view
- Create volume along surface of input parts without boundary extents The volume follows the surfaces of the selected parts resulting from Any system, part, or reference plane. The volume depth (normal to the view) is extended by the values of Volume growth into drawing and Volume growth out of drawing. The volume area (in the plane of the view) is not extended to (nor clipped by) the boundaries defined by Boundary (Block, volume, or assembly). The gathering range is extended by the values defined by Extend Into and Extend Out Of.

Example: Corrugated bulkhead

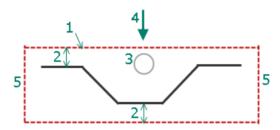


- 1 Cross-section of volume
- 2 Volume growth
- 3 Extended gathering range
- 4 View direction
- 5 Additional part gathered into view
- 6 Additional part not gathered into view

.

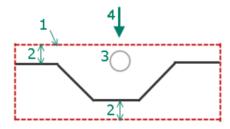
Create volume from range box around input parts with boundary extents - The volume is a rectangular range box around the selected parts resulting from Any system, part, or reference plane. The volume depth (normal to the view) is extended by the values of Volume growth into drawing and Volume growth out of drawing. The volume is extended to (or clipped by) the boundaries defined by Boundary (Block, volume, or assembly). See the corrugated bulkhead example below.

Example: Corrugated bulkhead

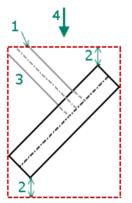


- 1 Cross-section of volume
- 2 Volume growth
- 3 Additional part gathered into view
- 4 View direction
- 5 Volume extended to boundaries
- Create volume from range box around input parts without boundary extents The volume is a rectangular range box around the selected parts resulting from Any system, part, or reference plane. The volume depth (normal to the view) is extended by the values of Volume growth into drawing and Volume growth out of drawing. The volume is not extended to (nor clipped by) the boundaries defined by Boundary (Block, volume, or assembly). See the corrugated bulkhead example below.

Example: Corrugated bulkhead



Example: Non-orthogonal member



- 1 Cross-section of volume
- 2 Volume growth
- 3 Additional part gathered into view
- 4 View direction

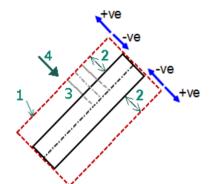
•

Create oriented volume around input parts without boundary extents - The volume is a rectangular box around the parts. The box is oriented around parts to minimize the volume. The parts are typically not orthogonal, but the view direction is typically normal to the box. The volume depth (normal to the view) is extended by the values of Volume growth into drawing and Volume growth out of drawing.

■ NOTES

- If the view direction is not normal to the box, then the volume is extended to (or clipped by) the boundaries defined by Boundary (Block, volume, or assembly) instead of Volume growth into drawing and Volume growth out of drawing.
- If the view direction is normal to the box, then the volume is extended to (or clipped by) Volume growth into drawing and Volume growth out of drawing. These values can be positive or negative.

Example: Non-orthogonal member



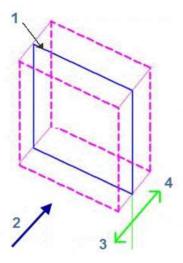
- 1 Cross-section of volume
- 2 Volume growth
- 3 Additional part gathered into view
- 4 View direction
- **Do not create volume** A volume is not used to restrict gathering using gathering rule. For example, a shell longitudinal profile view does not use a volume. It instead uses levels to gather secondary parts.

Depth Into

Defines a distance into the drawing to extend the depth of a thin volume. Parts within the thin volume are displayed in the view. A value for this query is optional.

Depth Out Of

Defines a distance out of the drawing to extend the depth of a thin volume. Parts within the thin volume are displayed in the view. A value for this query is optional.



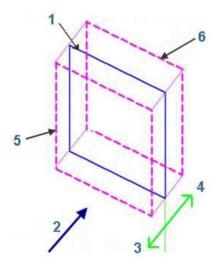
- 1 View plane
- 2 View direction
- 3 Out of drawing
- 4 Into drawing

Boundary Overlap

Defines the distance that parts extend past the view boundary.

Offset

Defines the distance from a reference plane used to create the view plane. A positive value offsets the view plane out of the drawing. A negative value offsets the view plane into the drawing. This value can be used in combination with **Volume growth into drawing** and **Volume growth out of drawing** to create a thin volume. A value for this query is optional.



- 1 Reference plane
- 2 View direction
- 3 Offset out of drawing (positive value)
- **4** Offset into drawing (negative value)
- 5 View plane offset out of drawing
- 6 View plane offset into drawing

Extend Into

Defines the distance that the gathering range box extends into the drawing. You can display parts that are outside of the thin volume and in the extended gathering range box. A value for this query is optional.

Extend Out Of

Defines the distance that the gathering range box extends out of the drawing. You can display parts that are outside of the thin volume and in the extended gathering range box. A value for this query is optional.

Clip to Volume

Clips part visibility in the graphic view to the boundaries of the volume.

Gather Objects Options

Gathering Rule

Defines the type of parts gathered in the drawing view. The rule provides a definition of the objects to be gathered and drawn in a view. Select a rule that supports the type of drawing view you want to create, such as Steel Order, Piping, Pipe Supports, or Assembly.

Include all objects in volume

Defines how gathered parts are included in the drawing view. Select to gather all parts in the volume even if they are not connected to the selected parts. Clear to gather only the selected parts and parts connected to the selected parts.

Show Objects Gathered

Highlights the gathered parts.

Orient View Options

Orientation Rule

Specifies the rule used to define the view orientation. Select options such as Global Coordinate System, Local Coordinate System, Scantlings for Plate/Profile, and Scantling By Plate Normal.

View Direction

Defines the direction of the view. The available options are defined by the selected orientation rule.

Up Direction

Defines the up direction of the view. The available options are defined by the selected orientation rule.

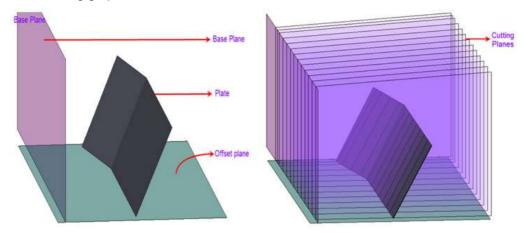
Orient Display to View Direction

Changes the orientation of the graphic view to match the view orientation. This option is only available when Local Coordinate System is selected for Orientation Rule.

For more information on view orientation options, see Orientation Rules in the Smart 3D Orthographic Drawings User's Guide.

Expansion View Ribbon

Provides options for generating expansion views. An overview of the required inputs is shown in the following graphic:



NOTE This ribbon displays only when the **Associate Views to Object Command** option is set to **Expansion**. For more information, see *General Tab (Edit Ruleset View Style Dialog Box)* in the *Drawings and Reports Reference Data Guide*. You can access this document using the **Help > Printable Guides** command in the Drawings and Reports task.

Expansion Surface

Select a plate system or plate part for which you want expansion drawings. You can select multiple plate systems or plate parts.

Cutting Planes Axis

Select a coordinate system axis for defining the cutting planes.

Base Plane Definition

Select a plane that is normal to the grid planes of the cutting plane axis.

Offset Plane

Select a plane that is perpendicular to the grid planes of the base plane and cutting plane axes. This plane is the reference plane from which the curve is expanded. This is an optional input.

Expansion Direction

Defines the expansion direction. This option is available only when the base plane intersects the expansion surface. If you select **Positive Direction**, the expansion surface on the positive side of the base plane will be expanded and its corresponding drawing will be generated. If you select **Negative Direction**, the expansion surface on the negative side of the base plane will be expanded and its corresponding drawing will be generated. If you select **Both**, the expansion surface will be expanded completely and its corresponding drawing will be generated.

☐ Block or Volume

Select a bounding block or volume. This is an optional input.

Cancel

Clears the selected options and allows you to continue the selection process.

Accept

Accepts the selected options.

Finish

Finishes the command.

For workflows using this command, see Associate Objects to Views (on page 390) and Associate Objects to a Drawings by Rule View (on page 395).

■ NOTES

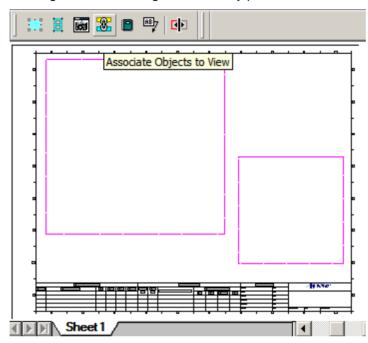
- When using a Snapshot view style, all objects within the associated volume are included in the drawing view, regardless of exclusions made using Associate Objects to View 35.
- When using a **Volume** view style, only the objects returned by the filters in the view style are included in the drawing view.
- An error message displays if a volume is not selected in the 3D application.
- If a 3D task process is running (for example: a recompute of relationships after moving a grid plane), an informational message displays and Associate Objects to View is canceled.
- Views that are too small to display the volume are automatically sized larger to fit unless the view is set to Fit to Scale or is managed by a region. Likewise, views that are larger than the volume are automatically resized to fit unless the view is set to Fit to Scale or is managed by a region. The view size grows or shrinks from the center of the view and view proportions may change after the resize. The drawing must be saved to make the resize of the view permanent. For more information, see Automatic Resize Behavior of Composed Views (on page 358).
- To remove associated inputs from a view, use Remove Associated Inputs . For more information, see Remove Associated Inputs Command (on page 396).
- Section and detail views inherit filters used by the Filter option from the parent view.

See Also

Place View Command (on page 347) Update View Command (on page 485)

Associate Objects to Views

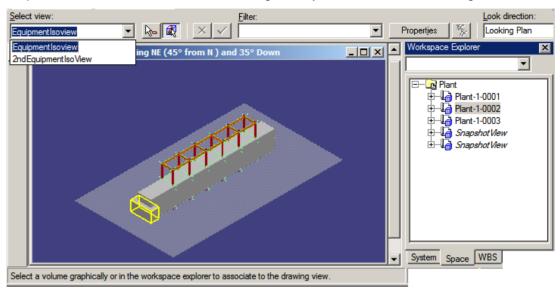
The following procedure steps you through associating volumes to drawings views in a 3D composed drawing. The procedures assumes you have already opened a 3D drawing in SmartSketch Drawing Editor to place and associate views. The example workflow uses a drawing with two drawing views already placed.



■ NOTE For more information on 3D composed drawings, see the *Orthographic Drawings* User's Guide. For information on the commands available for editing 3D drawings, see *Working with Drawings and Reports and SmartSketch Drawing Editor* (on page 321).

Associate Drawing Views with Orthographic View Styles

- 1. In SmartSketch Drawing Editor, select the orthographic drawing views that you want to associate, and then click **Associate Objects to View** 38.
- In the 3D application, notice the Associate Objects to View ribbon above the active window. If the selected drawing view has an orthographic view style, the Select View dropdown includes the names of the drawing views you selected in the drawing.

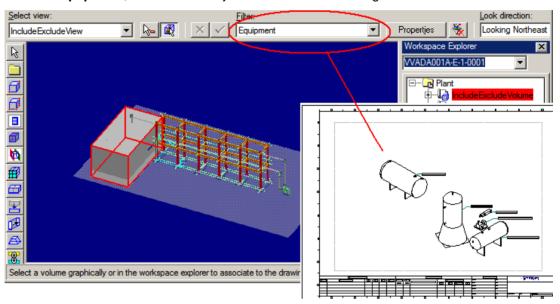


- 3. Click volume graphically or in the **Workspace Explorer** to associate to the current drawing view. The **Select View** dropdown scrolls automatically to the next drawing view in the list. Click another volume to associate with the second drawing view.
- 4. When you are done associating volumes to drawing views, return to SmartSketch Drawing Editor, and click **Place View**
- 5. To update the view contents with the associated volumes, select and right-click the view and select **Update View** on the shortcut menu. For more information, see *Update View Command* (on page 485).
- 6. Save the drawing document and exit SmartSketch Drawing Editor.
- 7. Update and complete your drawing document.

Exclude Objects from the Volume

The **Associate Objects to View** command ribbon allows you to further define the objects associated to the drawing view. You can use filters to restrict the objects collected from the volume. You can also exclude objects from the volume if you do not want them included in the drawing.

In the following graphic, the **Associate Objects to View** ribbon shows how you can use a filter, such as **Equipment**, to define the objects shown in the drawing.



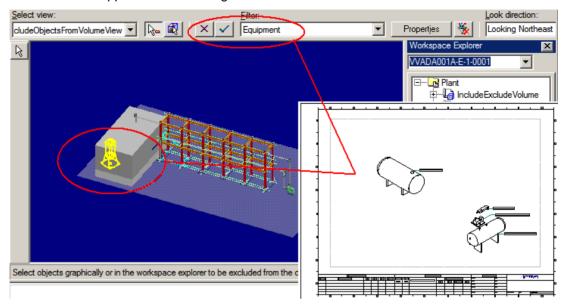
In the next example, the same **Equipment** filter is applied to the volume. However, a **Vertical Tank** is excluded from the filter. You achieve this output as follows:

- 1. Select the **Filter**, then click **Exclude Objects**
- 2. Select the objects inside the volume that you want to exclude from the drawing.

■ NOTES

- We recommend that you associate your volume before excluding objects from the volume.
- You can select the objects from the model, or from the Workspace Explorer, but you
 cannot select referenced objects that originate from files on the Reference tab of the
 Workspace Explorer.
- You can clear objects that have previously been excluded from the drawing view by clicking on the object.
- If you exclude an object while the related tab of the **Workspace Explorer** is active, the child objects are automatically excluded. For example, if the **System** tab is active when you exclude a system object, then all of the children objects are also excluded. If the related WSE tab is not active, only the object that you specifically select is excluded. For example, if you exclude a pipe run while the **Space** tab is active, the child parts of the pipe run are not excluded.

- 3. Click **Accept** ✓ to save the change to the volume definition.
- 4. Go back to SmartSketch Drawing Editor and update the view contents with the associated volumes with the **Update View** command on the shortcut menu of the view. The **Vertical Tank** does not appear in the drawing view.



- 5. Save the drawing document and exit SmartSketch Drawing Editor.
- 6. Update and complete your drawing document.

Associate Key Plan or Report Views

- 1. Select a key plan or report view in the drawing document.
- 2. Click Associate Objects to View 38.
- 3. In the 3D application, select a view from the **View** dropdown. The **View** dropdown contains a list of all the drawing views available in the current drawing document open in SmartSketch Drawing Editor.
- 4. Click **Finish** to save the association to the key plan or report view.
- 5. Go back to SmartSketch Drawing Editor to update the view contents. Select and right- click the view and select **Update View** on the shortcut menu. For more information, see *Update View Command* (on page 485).
- 6. Save the drawing document and exit SmartSketch Drawing Editor.
- 7. Update and complete your drawing document.

Associate to a Drawing View with a Ruleset View Style

Ruleset view styles are only used in Smart 3D drawings.

- 1. Select a drawing view with a ruleset view style then click **Associate Objects to View 38**.
- 2. In the 3D application, select a query from the **Query** dropdown. Depending on the query, you may need to specify a value or select objects graphically.
- 3. Click **Finish** to save the association to the drawing view.
- 4. Go back to SmartSketch Drawing Editor to update the view contents. Select and right- click the view and select **Update View** on the shortcut menu. For more information, see *Update View Command* (on page 485).
- 5. Save the drawing document and exit SmartSketch Drawing Editor.
- 6. Update and complete your drawing document.

See Also

Place View Command (on page 347)
Associate Objects to View Command (on page 374)

Associate Objects to a Drawings by Rule View

- 1. In Smart 3D, switch to a task other than Drawings and Reports, such as Molded Forms or Structural Detailing.
- 2. Click Tools > Drawing Console.

The **Drawing Console** dialog box displays.

- 3. Right-click an existing Drawings by Ruleset drawing document and click **Edit**.
 - SmartSketch Drawing Editor displays.

The Drawing View Properties Dialog Box displays.

5. On the **View** tab, select **More** for the **Style** property.

The **Select View Style** dialog box displays.

6. Navigate through the hierarchy and select a drawing view style, and then click **OK**.

Additional properties display on the View tab of the Drawing View Properties dialog box.

- **NOTE** For more information about ruleset view style definitions, see *Define View Style Command* in the *Drawings and Reports Reference Data Guide*. You can access this document using the **Help > Printable Guides** command in the Drawings and Reports task.
- 7. On the **View** tab, type a value for **Name**, and then select the needed values for **Coordinate System**, **Scale Family**, and **User Selected Scale**.
- 8. Click OK.

- 9. Select the graphic view in the drawing area and click **Associate Objects to View** 38.
- 10. Select the 3D application window to make it the active window.
- 11. Use the ribbon that displays in the 3D window to define the necessary options.

■ NOTES

- For more information, see Place View Command (on page 347) and Associate Objects to View Command (on page 374).
- The software displays either the **Drawing Ruleset Ribbon** or the **Expansion Ribbon**, depending on how you defined the **Associate Objects to View** options for the selected view style. For more information about this option, see *General Tab (Edit Ruleset View Style Dialog Box)* in the *Drawings and Reports Reference Data Guide*.

Remove Associated Inputs Command

Allows you to remove the associated volumes, filters, views, and other inputs from the selected view. It works with all graphic and non-graphic (report and keyplan) views. The command is available on the Compose toolbar when creating a new 3D composed drawing or editing a 3D composed drawing.

When you click **Remove Associated Inputs**, a status bar message instructs you to select a single drawing view. After you select a drawing view, a confirmation message displays. Click **Yes** to continue and remove the associated inputs from the selected view, or click **No** to cancel the command.

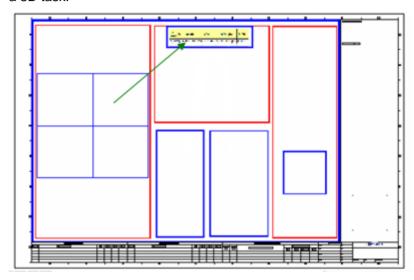
NOTE You cannot **Undo** or **Redo** actions performed by the **Remove Associated Inputs** command.

See Also

Place View Command (on page 347) Associate Objects to View Command (on page 374) Update View Command (on page 485)

Place Report View Command

Adds a report "view" in the drawing. The report view becomes an embedded report based on the contents of a drawing view. You could create Bill of Material, Issue Record, or a Revision report tied to the contents of a specified drawing view. This command is only available in **SmartSketch Drawing Editor** when you create a new drawing or open an existing drawing from a 3D task.



When you click **Place Report**, you click and drag to define the new report view. The **Report Properties** dialog box appears so you can define the properties associated with the report view.

When you select a report view and click **Associate Objects to View** , the **Associate Objects to View** ribbon appears on the 3D task window. For more information, see *Associate Objects to View Command* (on page 374).

For information on creating new views, see *Place View Command* (on page 347).

For a list of reports that can be embedded in a drawing successfully, see *Reports Compatible* with *Drawings* (on page 399).

■ NOTES

- You can associate multiple reports to the same view, but you cannot associate multiple views to the same report.
- You cannot multi-select report views to associate to a single view. The relationship is one-to-one from report view to graphic drawing view. Also, you cannot multi-select report views and graphic views. Other objects (such as lines or symbols) are ignored if they are in the select set with the report view when you run the Associate Objects to View command.

See Also

Report Properties Dialog Box (Place Report View Command) (on page Update View Command (on page 485)

Report Properties Dialog Box (Place Report View Command) (on page

Report Properties Dialog Box (Place Report View Command)

Specifies properties for the selected report view. This dialog box displays after you have placed a new report view using the **Place Report View** command or when you edit properties for an existing report view.

Name

Specifies a name for the report view.

Description

Provides a description of the report view.

Report Template

Specifies the report template to use when populating the report view. Select **More** in the **Value** list to display the **Select Template** dialog box. For more information, see *Select Template Dialog Box* (on page 398).

Report Template Output

Specifies how the report output will be formatted. Select **Excel** to output the report as an Excel spreadsheet or **Native text boxes** to use a text box format. The **Undefined** value means the software uses the default output format for the report (Excel spreadsheet).

Report Justification

Specifies the justification of the report window. Select **Top-Left**, **Top-Right**, **Center-Center**, **Bottom-Left**, or **Bottom-Right** to align the report to one of these positions. For example, if you select **Top-Right**, the top-right corner of the report window is aligned to the top-right corner of the view. The **Center-Center** option is the only justification option that scales the report window to fit the view. The other justification options do not scale to the view. For example, if you choose **Top-Right** and the report window is either larger or smaller than the view, it is not resized automatically to fit the view.

■ NOTE The Report Output Format setting must be set to Native text boxes for justification to function. Otherwise, the justification setting is automatically set to Center-Center.

After creating the view and defining its properties, use the **Associate Objects to View** command to specify the content of the view.

The **SP3DConvertExcelEmbedded.dll** is a delivered custom command that allows you to convert an Excel spreadsheet report to the native text box format for use in 3D Drawings. For information on converting Excel spreadsheet reports, see *Convert Excel Spreadsheet Reports to Native Text Box Format Custom Command* (on page 491).

See Also

Place Report View Command (on page 396)

Select Template Dialog Box

Specifies a template to associate with the selected view.

This dialog box appears when you click **More** in the **Value** dropdown on the **Report Properties** dialog box while placing a new report view on the drawing sheet. For more information, see *Place Report View Command* (on page 396).

Click **OK** to return to the **Report Properties** dialog box.

See Also

Place Report View Command (on page 396) Report Properties Dialog Box (Place Report View Command) (on page

Reports Compatible with Drawings

There are three conditions that must be satisfied before a report can be successfully embedded in a drawing and produce useful results.

- Pure SQL queries are not supported
- Report must have a column named OID and LocatableOID
- There must be a column named ItemNum

The following reports can be embedded in a drawing successfully. The product allows you to embed any of the delivered reports in a drawing, but the following reports provide more meaningful results when embedded in a drawing. The following reports are the only reports that are compatible with the bubble label (Reference_Circle_CA_L) in drawings. For more information on each report, see the *Smart 3D Reports User's Guide*.

PipeRuns by Drawing (Deliverables\PipeRuns by Drawing)

Equipment Material Take-Off (Equipment)

Solids of Design Equipment with Composite CoG (Equipment)

Hangers and Supports for Components - Drawings (Hangers and Supports)

Hangers and Supports for Supported Route - Drawing (Hangers and Supports)

Hangers and Supports for Supporting Structure - Drawing (Hangers and Supports)

Hangers and Supports End View - Drawing (Hangers and Supports)

Hangers and Supports ISO View - Drawing (Hangers and Supports)

Hangers and Supports Side View - Drawing (Hangers and Supports)

Hangers and Supports Pipe Rack Sorted by Name - Export to Icarus (Hangers and Supports)

Equipment Piping Trim (Piping)

HVAC Material Take-Off (HVAC)

Piping Fittings - Export to Icarus

Piping Material Take-Off (Piping)

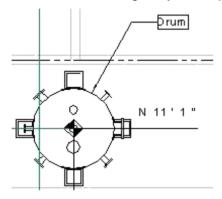
Piping Pipe Line List (Piping)

Place a Label Command

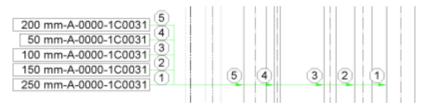
Manually labels items on a drawing. This command is available when you are editing a 3D drawing in SmartSketch Drawing Editor.

You may want to manually place labels to identify or emphasize objects beyond what is provided by automatic labeling. You can choose one of the delivered label rules, or you can customize your own label rule.

You can select a single object and place a label for the object.



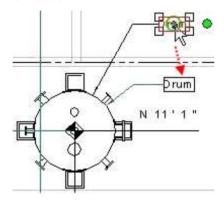
With an applicable label rule, you can also select multiple objects, and then place a group of labels. The following figure shows grouped labels aligned to each other and consolidated by label types with optional leaders and bubble labels. For more information, see *Group Labels* (on page 405).



When you place labels using this command, the software automatically saves the labels to the **DwgTemplate** layer so they are preserved when you update the drawing document.

TIPS

To move a label after it has been placed, click and drag the label by its origin point, identified by the green circle in the middle of the label object. In the following example, the **Drum** label is moved.



Snapping behavior is automatic. In the previous example, if you click and drag the **Drum** label by its origin point and hover over another label, then release the **Drum** label, it snaps to the new position.

Place a Label Ribbon (on page 401)

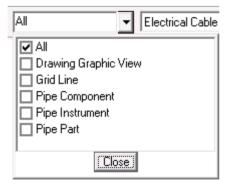
Place a Label Ribbon

Sets options for manual label placement on a drawing. To access this ribbon, click **Place a Label** .

■ NOTE When you place labels using this command, make sure you are on the **DwgTemplate** layer of the drawing so the software preserves the manually placed labels when you update the drawing.

Object Selection Filter

Specifies what type of objects can be selected in the drawing when placing a label. The list displays only object types in the drawing. Select one or more object types, or select **All** to select all object types. Click **Close** to close the list. **All** is the default value.



Label Name

Specifies a label rule. The label rule defines the label type:

- A single or a group label
- Inclusion and type of leaders
- Inclusion and type of bubble labels

Label rules are located in the [Reference Data Folder]\SharedContent\Drawings\Catalog\Rules\LabelRules folder.

TIP Click More to select a label rule from a hierarchical list.

Font

Defines the text font in the label.

Text Size

Defines the text size in the label.



Defines the text color in the label.

B Bold

Applies bold formatting to the label text.

I Italic

Applies italic formatting to the label text.

□ Underline

Underlines the text.

⋉- Shape

Defines a shape for the label border.



#BC / ∰ Orientation

Specifies the positioning and direction of text in the label. Select **Horizontal** \longleftrightarrow or **Vertical**

Leaderline On/Off

Specifies whether you want a leader line pointing to the object.

Leaderline Jog On/Off

Specifies whether a jog in the leader line is acceptable.



Defines the color of the leader line. The leader line color can be different than **Text Color**



Specifies whether the leader terminator originates inside or at the boundary of the object. Select **Boundary** or **Inside**.

Tolerance Zone

Specifies the required distance between the labeled object and the label before a leader appears. For example, if the leader tolerance value is set to 10 mm, then the label must be at least 10 mm away from the labeled object before a leader appears.

L*1 / 🖟 / 📂 / 😽 Dimensioned Label

Places a label in a dimension format. Choose from **Distance Between First**, **Coordinate Dimension**, **SmartDimension**, and **Angle Between**. A label displays instead of a dimension value. For more information, see *Dimensioned Label Command* (on page 414).

Pipe BOP

Places a label that displays the elevation or inverted elevation value of a pipe route object. For more information, see *Elevation Label Command* (on page 416).



Specifies that the label be shown using default values in the label rule. The software ignores font, text size, text color, and other formatting options.

Place a Manual Label

- 1. Right-click a drawing, and click **Edit** on the shortcut menu. The drawing opens for editing in **SmartSketch Drawing Editor**.
- 2. On the **Drawings Compose** toolbar, click **Place a Label**
- Click Object Selection Filter and select the types of objects available for selection. For example, selecting Physical Connections only allows you to select physical connections. Click Close.

- 4. Select an object in a drawing view to label. If you are placing a group label, select two or more objects in a drawing view.
- 5. In the Label Name box, select a label rule.
 - Select a label rule applicable to the selected objects.
 - To pick from a hierarchical list:
 - i. Select More.

The Select Label dialog box displays.

- ii. Select a label template folder in the left pane of the dialog box, select a specific label name in the right pane, and then click **OK**.
- With an applicable label rule, you can place a group label. For more information, see Group Labels (on page 405).
- 6. Make the required typeface selections in the Font, Text Size, and Text Color boxes
- 7. Click **Bold B**, **Italic** *I*, and **Underline** <u>u</u> to achieve the required formatting.
- 8. Click **Shape S**-, and select the shape of the label border.
- 10. Make the required leader line selections in the Leaderline On/Off /, Leaderline Jog On/Off /, and Leader Color boxes.
- 11. Click **Boundary** to clip the leader at the object boundary, or click **Inside** to extend the leader to the inside of the object.
- 12. Type a value in the **Tolerance Zone** box. This value determines the distance required for a leader to display.
- 13. Select a dimension label [**] in order to place a dimension-like label. For more information, see *Dimensioned Label Command* (on page 414).
- 14. Select an elevation label if you want to place a label that displays the elevation of a piping object. For more information, see *Elevation Label Command* (on page 416).
- 15. Select **As Drawn** if you want the label appearance to reflect the label rule definition only, ignoring any formatting overrides.
- 16. Click the drawing to place the label.
 - TIP During placement, the label will automatically align to the left or right side of an existing label. You can also align the label to any object keypoints in the drawing. For more information on using alignment settings, see Align Dimensions Command in the SmartSketch Drawing Editor Help.

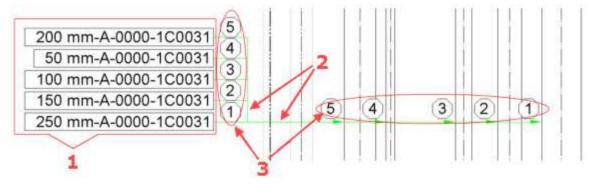
- 17. Continue placing labels on the drawing if required.
- 18. Right-click to end the command.
- 19. Save your changes before exiting SmartSketch Drawing Editor.

NOTE You can remove manual edits from a drawing using **Clear Manual Edits** in **SmartSketch Drawing Editor**. For more information, see *Clear Manual Edits Command* (on page 462).

Group Labels

■ NOTE The software delivers **Name_None_CA_JL** as an example label rule that places single or group labels. See your administrator for other customized group label rules that may be in your catalog.

A group label allows selection of multiple objects and groups the labels for the objects. Optionally, the group label also places leaders and bubble labels.



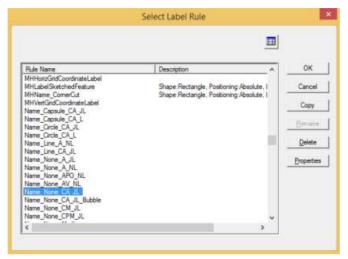
- 1 Group label
- 2 Leaders
- 3 Bubble labels

You can add a group label as an option to any manually placed label. The **Place a Label** command places a single label when you select one object or a group label when you select multiple objects. The software delivers the **Name_None_CA_JL** label template as an example implementation of single and group labels. You can find the Name_None_CA_JL.xml file in the [Reference Data Folder]\SharedContent\Drawings\Catalog\Labels\Templates folder.

Defining a group label

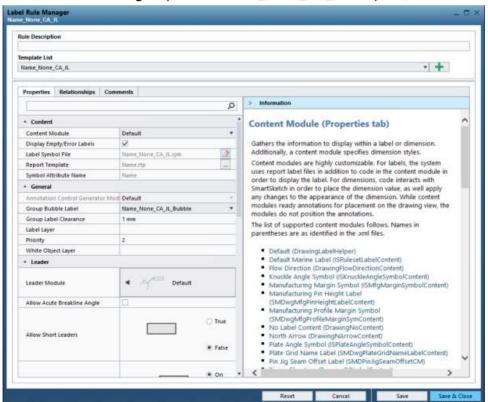
■ NOTE You must have already defined the view style type and the **View Style Properties** dialog box must be displayed. For more information, see *Define View Style Dialog Box* and *View Style Properties Dialog Box*.

1. Select Name_None_CA_JL in the Select Label Rule dialog box, as shown below.



2. Click Properties.

The Label Rule Manager opens the Name_None_CA_JL template, as shown below.



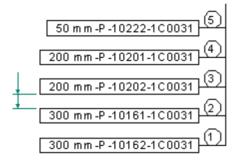
- 3. To change the distance between grouped labels, type a new value in the **Group Label Clearance** box.
- 4. To suppress the bubble the software draws around grouped labels, select **NONE** from the **Group Bubble Label** list.
- 5. Click Save & Close to exit the Label Rule Manager.

Group Bubble Label

Defines the bubble label template and symbol used by the group label. If you select **NONE**, then the software places the group labels without bubble labels.

Group Label Clearance

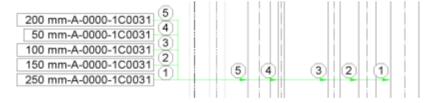
Specifies the distance between labels in the group.



Leaders

Leaders jog as needed based on the position of the label group to the objects, as shown in the following examples.

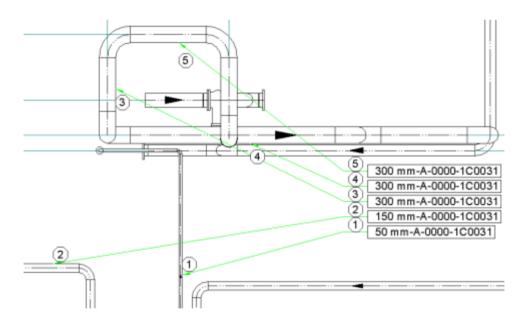
Group label with horizontal leader



Group label with vertical leader



Group label with fanned leader



Group label with no leader

Duct	Banks_	_Conduit	Run-1-41
Duct	Banks_	Conduit	Run-1-42
Duct	Banks	Conduit	Run-1-43
Duct	Banks	Conduit	Run-1-44
Duct	Banks_	Conduit	Run-1-45
Duct	Banks	Conduit	Run-1-36
Duct	Banks	Conduit	Run-1-37
Duct	Banks	Conduit	Run-1-38
Duct	Banks	_Conduit	Run-1-39
Duct	Banks_	Conduit	Run-1-40
Duct	Banks_	_Conduit	Run-1-31
Duct	Banks	Conduit	Run-1-32
Duct	Banks_	Conduit	Run-1-33
Duct	Banks	Conduit	Run-1-34
Duct	Banks_	Conduit	Run-1-35

Sorting and Alignment

The software sorts the labels according to the relative location of the objects (as defined by the object center points) and the type of leaders.

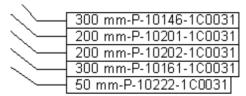
Leader Type	Sorting		
Horizontal	Left to right		
Vertical	Top to bottom		
Fanned	To minimize crossing of leaders		
None	Top to bottom or left to right		

Labels align to the leader connection side.

Right side alignment

50 mm-P-10222-1C0031	/
300 mm-P-10146-1C0031	
300 mm-F-10140-1C0031	
200 mm-P-10201-1C0031	
200 mm-P-10202-1C0031	
300 mm-P-10161-1C0031	

Left side alignment



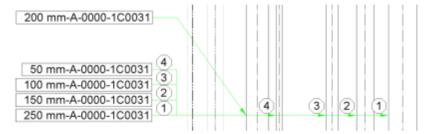
Selecting multiple objects for the group label

Use one of the following methods to select objects in the SmartSketch Drawing Editor.

- Press CTRL and select each object individually in a drawing view.
- Click and drag in a drawing view to create a selection window. The software selects all objects completely within the selection window.
- Press CTRL and then click and drag in a drawing view to create a line selection. The software selects all objects overlapping the line.

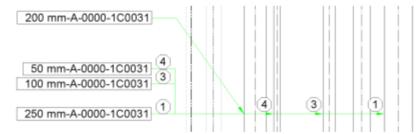
Moving a label in a group

You can select and move a label to separate it from the group label.



Deleting a label in a group

You can select and delete a label. The software also deletes the leader and bubble labels, but the remaining bubble labels are not renumbered.



Limitations

Group labels have the following label parameter limitations:

- Orientation () is ignored. Group label text is always horizontal.
- **Boundary 🗖 / ₫** is ignored.
- Tolerance Zone is ignored.

The following values for **Shape** cannot be used as the label border: **Cloud** \bigcirc , **Pentagon** \bigcirc , **Triangle Up** \triangle , and **Triangle Down** ∇ .

For more information on label parameters, see *Place a Label Ribbon* in *Place a Label Command* (on page 400).

Place a Manual Weld Label

- 1. Right-click a drawing, and click **Edit** on the shortcut menu. The drawing opens for editing in **SmartSketch Drawing Editor**.
- 2. On the toolbar, click Manually Place Labels **.
- 3. (This step is only for marine mode) Use the **Object Filter** to enable only certain objects for selection. For example, selecting **Physical Connections** in the Object filter list only allows you to select physical connections.
- 4. In the Label Name drop-down menu, select a label rule.
 - TIP The label rules are located on the application server in the \Symbols\Drawings\Catalog\Labels\Templates folder.
- 5. To pick from a hierarchical list, select More.

The **Select Label** dialog box displays.

- 6. Select a label template folder in the left pane of the dialog box, select a specific label name in the right pane, and then click **OK**.
- 7. Click on a physical connection.

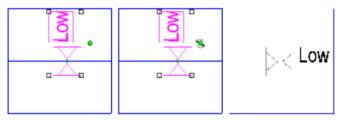
The correct weld symbol automatically displays based on the physical connection properties.

- 8. In the **Font** box, specify the font.
- 9. In the **Text size** box, specify the size of the text.
- 10. Click **Text Color** and select a color for the label text.
- 11. Click the **Bold B**, **Italic** *I*, and **Underline U**buttons to achieve the necessary formatting.
- 13. Select a leader and leader jog.
- 14. Click **Leader Color** and select a color for the leader.
- 15. Choose the **Boundary** option to clip the leader at the object boundary, or choose the **Inside** option to extend the leader to the inside of the object.
- 16. Key in a value for the **Leader Tolerance** value. This value determines the distance required for a leader to display. For example, if the leader tolerance value is set to 10mm, the label must be at least 10mm away from the labeled object before a leader displays.
- 17. Choose a dimension label in order to place a dimension that acts as a label. For more information, see *Dimensioned Label Command* (on page 414).

18. Select **As Drawn** if you want the label appearance to reflect the label definition only and not have any formatting overrides.

TIPS

- If you select As Drawn, the label appearance is determined by the label definition. You cannot set the font, text size, text color, and so forth for the label.
- Click the drawing to place the label. If you hover directly over the physical connection, the label will snap to the physical connection line, but will not align.
- To move the label after it has been placed, click and drag the label by its origin point, identified by the green circle in the middle of the label object. In the graphic below, the weld symbol is moved.
- To rotate the label after it has been placed, select the label and click the green circle located on the outside of the label. Drag the green circle until the label is aligned correctly to the physical connection.



- 19. Continue placing labels on the drawing if necessary.
- 20. Right-click to end the command.
- 21. Save your changes before exiting **SmartSketch Drawing Editor**.

NOTE You can remove manual edits from a drawing using **Clear Manual Edits** in **SmartSketch Drawing Editor**. For more information, see *Clear Manual Edits Command* (on page 462).

See Also

Place a Manual Label (on page 403)

Place a Manual Flow Arrow on an Orthographic Drawing

You can place flow arrows on pipe objects in orthographic drawings with the **Place a Label** command in **SmartSketch Drawing Editor**.

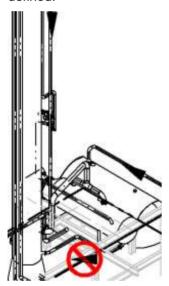
You can also add flows arrows as part of the drawing view style definition. For more information, see *Add Flow Arrows to Orthographic Drawings* in the *Drawings and Reports Reference Data Guide*.

- 1. Right-click a drawing, and click **Edit** on the shortcut menu. The drawing opens for editing in **SmartSketch Drawing Editor**.
- 2. On the toolbar, click Manually Place Labels
- 3. Click an object in a drawing view to label.
- 4. In the **Label Name** drop-down menu, select **More** to display the **Select Label** dialog box.

- 5. Select a flow label template folder in the left pane of the dialog box, select a specific label name in the right pane, and then click **OK**.
- 6. Select a pipe object in the drawing to place the label.



If the arrow has a red-crossed circle over it, the selected pipe does not have flow direction defined.



- 7. Right-click to end the command.
- 8. Save your changes before exiting **SmartSketch Drawing Editor**.

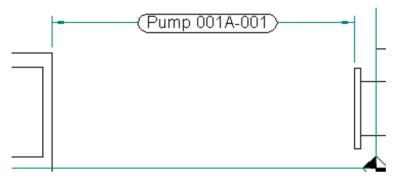
■ NOTES

- To move the flow arrow after it has been placed, click and drag the arrow by its origin point, identified by the green circle in the middle of the flow arrow object.
- You can remove manual edits from a drawing using Clear Manual Edits in SmartSketch Drawing Editor. For more information, see Clear Manual Edits Command (on page 462).
- Centerlines must display for pipelines before flow arrows can be placed.

Dimensioned Label Command

Places a label with dimension formatting. The dimension displays the label instead of a dimension value. **Dimensioned Label** is an option on the **Place a Label** ribbon. You can select **Distance Between** ; Coordinate **Dimension**, SmartDimension for Angle Between Command, Coordinate Dimension Command, SmartDimension Command, and Angle Between Command in the SmartSketch Drawing Editor Help.

Before selecting **Dimensioned Label**, you must select an appropriate label from **Label Specification** on the **Place a Label** ribbon. The following example shows a dimensioned label on a composed drawing. A **Distance Between** dimension is placed between two pumps with the **Equipment_Plane Equipment_Name** label selected.



NOTE Only two objects can be selected with **Dimensioned Label**. The label placed is determined by the first object you select, not the second.

See Also

Place a Dimensioned Label (on page 415) (Marine mode only)
Place a Dimensioned Label (on page 414) (Plant mode only)

Place a Dimensioned Label

This workflow places a dimensioned label on a composed drawing.

- Right-click a drawing, and click Edit on the shortcut menu.
 The drawing opens in SmartSketch Drawing Editor.
- 2. On the toolbar, click Manually Place Labels.
- 3. In the Label Name drop-down menu, select the Structural Framing Elevation_Grid Line_TOS label from the delivered folder.
 - **NOTE** To pick from a hierarchical list, select **More**.
- 4. Select Coordinate Dimension from the dimension label drop-down menu. The dimensioned label ribbon displays.
 - NOTE You can select any of the dimension commands from the drop-down menu.
- 5. Select a structure member to label.

6. Click to place the Coordinate Dimension label.



- TIP You can use custom dimension styles when placing dimension-like labels. For more information, see *Dimension Properties Dialog Box* in the *SmartSketch Drawing Editor User's Guide*.
- 7. Continue placing labels if necessary.
- 8. Right-click to end the command.
- 9. Save your changes and exit SmartSketch Drawing Editor.

See Also

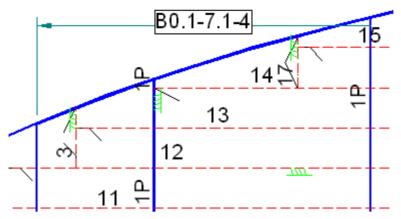
Dimensioned Label Command (on page 414) Place a Manual Label (on page 403)

Place a Dimensioned Label

This workflow demonstrates the ability to place a dimensioned label on a marine mode Steel Order drawing by placing a dimension-like label on two edges of a plate. For more information on dimensioned labels, see *Dimensioned Label Command* (on page 414).

- Right-click a Steel Order drawing, and click Edit on the shortcut menu. The drawing opens in SmartSketch Drawing Editor.
- 2. On the toolbar, click **Place a Label**
- 3. Use the **Object Filter** to enable only **Plate Parts** for selection.
- In the Label Name drop-down menu, select the Block Parent label from the delivered Steel Order folder.
 - **NOTE** To pick from a hierarchical list, select **More**.
- 5. Select the **Distance Between** dimension label command from the drop-down menu. The dimensioned label ribbon displays.
- 6. Select the edge of a plate that you want to label.
 - **NOTE** The first selected object is labeled, not the second.
- 7. Select a second edge. For example, select the opposite edge of the plate you selected.
 - NOTE The second point only determines how long the dimension line will be.

8. After selecting the second object, the correct label will display.



• TIP You can use custom dimension styles when placing dimension-like labels. For more information, see *Dimension Paper Space Objects for 3D Drawings* (on page 338) in the *SmartSketch Drawing Editor User's Guide*.

- 9. Continue placing labels on the drawing if necessary.
- 10. Right-click to end the command.
- 11. Save your changes before exiting SmartSketch Drawing Editor.

See Also

Dimensioned Label Command (on page 414)
Place a Manual Label (on page 403)

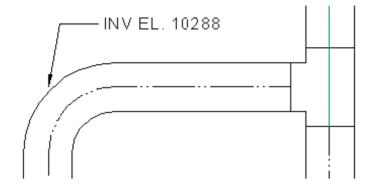
Elevation Label Command

Pipe BOP places a label that displays the elevation or inverted elevation value of the route object. For more information, see *Elevation versus Invert Elevation* below.

The **Pipe BOP** is available only if you select one of the following templates:

- Piping Plan_Routable_Center Elev_M Select to place a label for the center elevation of the pipe.
- Piping Plan_Routable_INV Elev_M Select to place a label for the inverted elevation of the pipe.
- Piping Plan_Routable_WPBOP Elev_M Select to place a label for the working point of the pipe.

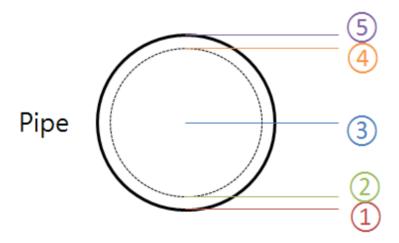
The following example shows an elevation label being used on a composed drawing. An inverted elevation is placed on a pipeline with the Piping Plan_Routable_INV Elev_M template selected.



NOTE Elevation labels support only manual placement and label templates for piping disciplines. To apply these templates to other disciplines, such as HVAC, cable tray, and conduit, modify and use these templates or create a new label template based on these templates.

Elevation versus Inverted Elevation

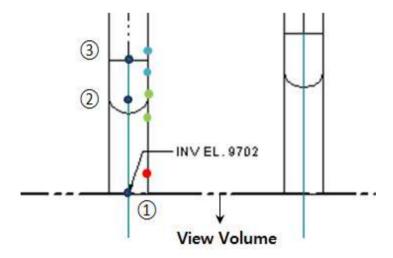
Inverted elevation is the inside elevation of a pipe, HVAC, cable, or conduit route object. Elevation and inverted elevation can be calculated at following locations:



Location INV (Inverted Elevation)		Elev (Elevation)			
Тор	4	Z of route object + (OutDiameter/2 - Thickness)		Z of route object + (OutDiameter/2)	
Center	3	Z of route object		Z of route object	
Bottom	2	Z of route object - (OutDiameter/2 - Thickness)		Z of route object - (OutDiameter/2)	

Leader Point Connection

When you click on the route object the leader point is automatically placed near port point, working point or clipping point, whichever is nearest to where you clicked. For example, if you click at the red point the leader point is moved to nearest clipping point (1). If you click at the green points, the leader point is moved to the nearest working point (2). If you click at the cyan points, then the leader point is moved to nearest port point (3).



Place an Elevation Label

This workflow places an elevation label on a composed drawing.

- 1. Right-click a drawing, and click **Edit** on the shortcut menu. The drawing opens in **SmartSketch Drawing Editor**.
- 2. On the horizontal toolbar, click Manually Place Labels.
- 3. In the **Label Name** drop-down menu, select the needed label. For example, to place an inverted elevation label select **Piping Plan_Routable_INV_Elev_M**.

The Pipe BOP command gets activated after you select the label template.

- **NOTE** To pick a label template from a hierarchical list, select **More**.
- 4. Select a route object to label.
- 5. Click on any point in the empty space to place the elevation label.
- 6. Continue placing labels if necessary.
- 7. Right-click to end the command.
- 8. Save your changes and exit SmartSketch Drawing Editor.

See Also

Elevation Label Command (on page 416)
Place a Manual Label (on page 403)

Group Selected Labels

Groups existing labels so that you can move them as a unit. You can choose to stack the labels directly above one another, or stack them diagonally. This simplifies the final formatting of labels that Smart 3D generates automatically.

Group Selected Labels Ribbon



Select by Labels

Indicates that the software selects labels to form the group. You can select the labels individually, by fence, or by dragging a line through the labels.



Select by Objects

Indicates that the software selects objects to form the label group. You can select the objects individually, by fence, or by dragging a line through the objects.

Labels Selection Filter

Specifies a filter to limit the labels you can select. The default option is All, which lets you select any label to add to your group. To refine the filter, select as many of the filters from the list as you need. You can select only those labels that match the filter.

Leaderline On/Off

Specifies whether you want a leader line pointing to the object.

Specifies whether a jog in the leader line is acceptable.



Leader Color

Defines the color of the leader line.

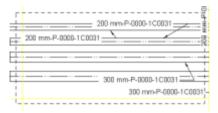
Selecting the Labels to Group

You have several options to select the labels to group. If you click **Select by Labels**, the software only lets you select labels. Click **Select by Objects** to select objects.

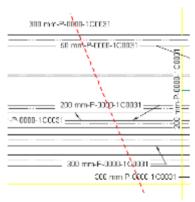
Press CTRL, and then individually click the labels or objects to group.



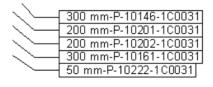
 Drag a fence around the labels or objects to group. The fence must start and stop in empty space.

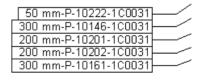


 Press CTRL, and then drag a line through the labels or objects to group. The line must start and stop in empty space.



The software places the group of labels without crossing the leader lines, even if the heights of the labels differ, and aligns the labels to the leader connection side.

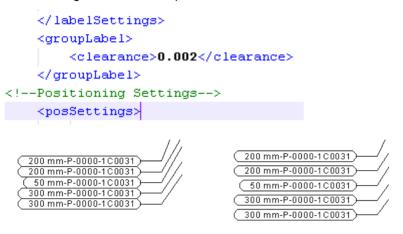




Left Right

The software sorts the labels so that no leaders in the group cross.

NOTE To change the distance between each label in the stack, your administrator can add a **<clearance>** tag to the label template.



Clearance value of 0.0

Clearance value of 0.002

Group existing labels

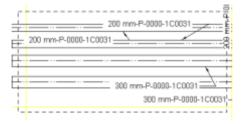
- 1. In the Drawings console, right-click a drawing, and click **Edit** on the shortcut menu.

 The drawing opens for editing in **SmartSketch Drawing Editor**.
- 2. On the **Drawings Compose** toolbar, click **Group Selected Labels**The **Label Filter** displays on the ribbon.
- 3. Click **Select by Labels** or **Select by Objects** to indicate how you want to make the selections for the group.
- 4. If necessary, select filters to refine the **Label Filter**, and then click **Close**.
- 5. Use the tools on the ribbon to specify the leader for the labels. For more information, see *Group Selected Labels* (on page 419).

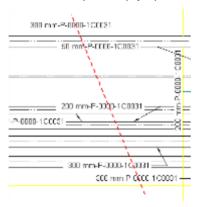
- 6. Select the labels or objects to group. You can use any of the following techniques:
 - Press CTRL, and then individually click the labels or objects to add to the group.



 Drag a fence around the labels or objects to group. The fence must start and stop in empty space.



• Press CTRL, and then drag a line through the labels or objects to group. The line must start and stop in empty space.

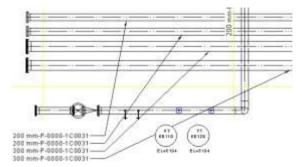


The software displays a blue preview outline to give you an approximation of the size of the stacked labels. The preview moves with the cursor.



7. If necessary, press CTRL to change the preview from stacked to diagonal.

8. Click the location for the relocated labels.

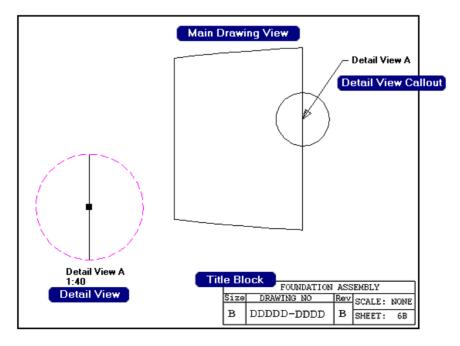


TIP To diagonally space the labels, press CTRL and click to indicate the location.



Place Detail Envelope Command

Creates a detail view for an existing drawing view. Detail views are more than enlargements of the main drawing view. They often contain additional graphical information that is not visible in the main drawing view, such as weld or chalk information. You specify the detail view by drawing a circle or polygon around a portion of the main drawing view. The shape created is called the *detail envelope*. The **Place Detail Envelope** command is only available in **SmartSketch Drawing Editor** when you create a new Composed drawing or open an existing Composed drawing from a 3D task. For example, the following graphic shows the main drawing view and a detail view:

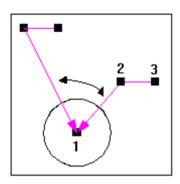


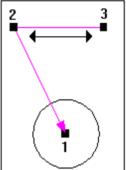
When you select an existing detail envelope, it highlights within the drawing view and the **Edit** ribbon appears so you can make modifications to the selected cutting plane. To place a detail envelope, select a drawing view and click the **Place Detail Envelope** command.

Callouts

Detail envelopes use callouts, or labels, that match the caption for the associated detail view. By default, the command places callouts in a fixed position on the main drawing view when you place the detail envelope. You can click-and-drag the callout to a new position, if necessary.

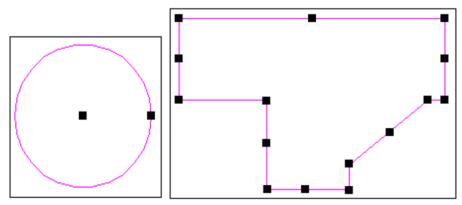
Callouts do not highlight when you select the detail envelope. You select callouts separate from the detail envelope.





Handles

Handles help you modify the shape and placement of the detail envelopes. Circle detail envelopes have two handles, a center point and an edge handle. The center point handle allows you to move the circle, while the edge handle allows you to change the size and shape of the circle. Polygon shapes have handles at each vertex. You can use the handles to move and change the shape of the polygon shape.



After you modify a detail envelope, click **Finish** to save the changes.

Delete Behavior

To delete a detail envelope, select it and press **Delete**. Deleting the detail envelope does not affect the drawing view. However, if there is an associated detail view, a message displays, asking if you want to convert the detail view to a normal drawing view or delete it with the detail envelope.

NOTE You can place detail envelopes inside section views and detail views.

See Also

Place Cutting Plane Ribbon (on page 428)
Place Detail Envelope Command (on page 423)
Place Detail Envelope Ribbon (on page 425)
Edit Detail Envelope Ribbon (on page 425)

Place Detail Envelope Ribbon

Sets options when placing a new detail envelope. This ribbon displays when you select a drawing view, then click **Place Detail Envelope** in the toolbar area.

If you are editing an existing detail envelope, the **Edit Detail Envelope** ribbon appears. For more information, see *Edit Detail Envelope Ribbon* (on page 425).

Circle

Specifies that you want to draw a round detail envelope shape.

C Polygon

Specifies that you want to draw a polygon detail envelope shape.

Reference mark

Displays the reference text below the detail view and on the detail view callout on the main drawing view.

Additional callout text

Provides a second line of text for the detail view callout.

See Also

Place Detail Envelope Command (on page 423)

Edit Detail Envelope Ribbon

Sets options when editing an existing detail envelope. This ribbon displays when you select a detail envelope on the drawing sheet, then click **Place Detail Envelope** in the toolbar area. If you have more than one detail envelope selected, this ribbon is disabled.

If you are placing a new detail envelope, the **Place Detail Envelope** ribbon appears. For more information, see *Place Detail Envelope Ribbon* (on page 425).

Reference mark

Displays the reference text below the detail view and on the detail view callout on the main drawing view.

Additional callout text

Provides a second line of text for the detail view callout.

NOTE The **Reference mark** and **Additional callout text** always show the current values for the detail envelope. The fields are disabled if the detail envelope is associated with a detail view.

See Also

Place Detail Envelope Command (on page 423)

Place a Detail Envelope

The following steps describe the basic workflow for placing a detail envelope on a drawing view.

- 1. In the 3D application, edit a drawing document. The drawing opens in **SmartSketch Drawing Editor**.
- 2. In the drawing document, select a drawing view. Zoom in on the area of the main drawing view to the location you want to place the detail envelope geometry.
- 3. In the toolbar area, click **Place Detail Envelope** . The **Place Detail Envelope** ribbon appears in the toolbar area.
- 4. Click **Circle** or **Polygon** △ so you can begin placing points to define the detail envelope. For more information, see *Place Detail Envelope Ribbon* (on page 425).
- 5. Click points as needed to create the detail envelope geometry. If you are creating a polygon shape for your detail envelope, complete the polygon by moving the cursor over the starting point until the **Close Polygon** symbol appears, then click to close the polygon.

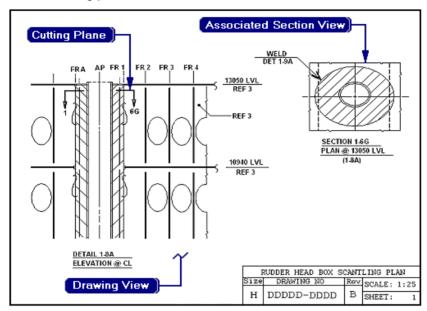
Edit a Detail Envelope

To edit an existing detail envelope, select the detail envelope. The **Edit Detail Envelope** ribbon appears so you can edit the settings on the detail envelope.

For more information on working with detail envelopes, see *Place Detail Envelope Command* (on page 423).

Place Cutting Plane/Section View Command

Creates a cutting plane on a drawing view. The cutting plane is an annotation marker that indicates where to slice a group of objects and from which direction to look at that slice. Cutting planes are made of one or more line segments. When comprised of multiple line segments, cutting planes are also referred to as "jogged" cutting planes. The **Place Cutting Plane** command is only available in **SmartSketch Drawing Editor** when you create a new drawing or edit an existing drawing from a 3D task. For example, the following graphic shows a drawing view, a cutting plane, and a derived section view:



You can only create section views from cutting planes drawn on valid graphical views. Valid graphical views include normal drawing views, section views, detail views, and snapshot views, but exclude keyplan views. Valid graphical views must already be associated to a volume in the model and updated at least once). If you delete a drawing view that has cutting planes, the cutting plans are also deleted. A confirmation message asks if you want to convert associated section views to normal drawing views or delete them as well.

After a cutting plane has been defined, a drawing view appears on your cursor. Click to place the view on the drawing sheet. For more information, see *Place a Cutting Plane/Section View* (on page 431).

When you select a cutting plane, it highlights within the drawing view and the **Edit** ribbon appears so you can make modifications to the selected cutting plane. To place a cutting plane, select a valid drawing view and click the **Place Cutting Plane** command. You can also click the command first and then select the view.

Single-segment and Jogged cutting planes have handles at the end-points, mid-points, and depth-points. You can change the geometry of the selected cutting plane by dragging its handles. Dragging a mid-point handle moves the associated line segment, maintaining its length and slope while altering any adjacent line segments. Dragging a vertex or end-point alters the length and/or slope of the attached line segment(s). Dragging a depth handle alters the depth of the associated section view.

You can move a single-segment cutting plane by clicking and dragging its mid-point handle to the new position. You can move a jogged cutting plane by clicking on a point of the line segment and dragging it; do not click a handle to move the jogged cutting plane.

After modifying a cutting plane, click **Finish** to save the changes. Selecting the **Update section** check box will cause an update of the related section view when the **Finish** button is clicked. The **Update section** option is remembered globally for all section views.

To delete a cutting plane, select it and press the DELETE key. Deleting the cutting plane does not affect the drawing view, but if there is an associated section view, you are asked if you want to convert the section view to a normal drawing view or delete it as well.

Shortcut Menu

When you select an existing cutting plane, you can right-click to get the cutting plane shortcut menu. The shortcut menu allows you to make the following modifications to the cutting plane: **Bring to Front**, **Send to Back**, **Pull Up**, and **Push Down**.

See Also

Place Cutting Plane Ribbon (on page 428)
Place a Cutting Plane/Section View (on page 431)
Section View Orientation Rules in Drawings by Rule (on page 498)

Place Cutting Plane Ribbon

Sets options when placing a cutting plane. This ribbon displays when you select a drawing view. Click **Place Cutting Plane** command to place a new cutting plane.

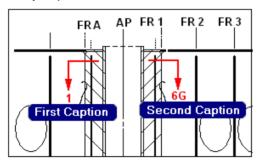
Reference 1

Displays the text shown for the first arrow of the cutting plane.

Reference 2

Displays the text shown for the second arrow of the cutting plane.

NOTE The **Reference 1** is always assigned to the point of the cutting plane that you placed first, while the **Reference 2** text is always assigned to the point of the cutting plane that you placed last.



Depth

Sets the depth of the cutting plane. This box only accepts positive values.

See Also

Place Cutting Plane/Section View Command (on page 426)
Place Section View Ribbon (on page 429)
Place a Cutting Plane/Section View (on page 431)

Place Section View Ribbon

Sets options when placing a section view. This ribbon displays after you have completed the placement of a cutting plane.

Update section

When checked, the section view updates after it is placed. This option is unchecked by default, but the last setting is saved as a session file preference.

View Style

Specifies a view style, which includes rules for filters, updates, and graphics. The view style controls the output characteristics of the view on the generated drawing. The list displays the 10 most recently used view styles in the session. Click **More...** to display the **Select View Style** dialog box.

NOTE Changing the view style for the section view can result in symbology different from that used in the parent view.

View Scale

Sets the scale family and scale used for the section view. The default scale setting is inherited from the parent view.

See Also

Place Cutting Plane/Section View Command (on page 426) Place Cutting Plane Ribbon (on page 428) Place a Cutting Plane/Section View (on page 431)

Edit Cutting Plane Ribbon

Sets options when editing a cutting plane. This ribbon displays when you select an existing cutting plane that is associated with a section view.

Finish

Saves the changes to the cutting plane. If the cutting plane is associated with a section view, the section view contents change if the **Update section** check box is selected.

Update section

When checked, the section view updates after it is placed. This option is not checked by default, but the last setting is remembered as a session file preference.

Reference 1

Displays the text shown for the first arrow of the cutting plane.

Reference 2

Displays the text shown for the second arrow of the cutting plane.

Depth

Sets the depth of the cutting plane. This box only accepts positive values.

See Also

Place Cutting Plane/Section View Command (on page 426) Place Cutting Plane Ribbon (on page 428)

Edit Section View Ribbon

Sets options when editing a section view. This ribbon displays when you select a section view.

Properties

Opens the **Drawing View Properties** dialog box, allowing you to change the properties of a drawing view. For more information, see *Drawing View Properties Dialog Box (Place View Command) - Steel Order Drawings* (on page 354).

Finish

Saves the changes to the section view.

Update section

When checked, the section view updates when you select **Finish**. This option is unchecked by default, but the last setting is remembered as a session file preference.

View Style

Specifies a view style, which includes rules for filters, updates, and graphics. The view style controls the output characteristics of the view on the generated drawing. The list displays the 10 most recently used view styles in the session. Click **More...** to display the **Select View Style** dialog box.

NOTE Changing the view style for the section view can result in symbology different from that used in the parent view.

View Scale

Sets the scale family and scale used for the section view. The default scale setting is inherited from the parent view.

Rotate

Rotates the section view clockwise by multiples of 90 degrees. When the view is rotated, user-placed and user-modified dimensions in the section view are deleted.

NOTE To rotate a section view, you must bulkload the following files to the database:

- [Product Folder]\CatalogData\BulkLoad\AdditionalDataFiles\Delta2009.1to2011\1_AMD_Delta_2 009.1 2011 Drawings.xls
- [Product Folder]\ShipCatalogData\BulkLoad\AdditionalDataFiles\DeltaSMV2011toV2011SP1\1_A MD_Delta_V2011_2011SP1_SM_ShipDrawings.xls. (Marine mode only)

See Also

Place Cutting Plane/Section View Command (on page 426)
Place Section View Ribbon (on page 429)

Place a Cutting Plane/Section View

Follow the general steps below to place a cutting plane on a drawing view.

- In a 3D application, edit a drawing document. The drawing opens in SmartSketch Drawing Editor.
- 2. In the drawing document, select a drawing view.
- 3. In the toolbar area, click **Place Cutting Plane** •• The **Place Cutting Plane** ribbon appears in the toolbar area. For more information, see *Place Cutting Plane Ribbon* (on page 428).
- 4. On the **Place Cutting Plane** ribbon, type the names for the reference text string you want shown on the cutting plane.
- 5. Click the first point in the line string, then proceed selecting as many points as you need to define the cutting plane geometry.
 - **NOTE** Jogged cutting planes are not supported in Ruleset drawings.
- 6. Right-click when you have finished defining the cutting plane line segments and are ready to define the depth of the cutting plane.
- 7. To change the view direction, move the cursor to the appropriate side of the cutting plane. The direction automatically changes according to which side of the cutting plane segment the cursor is on.
- 8. Click to define the section depth and view direction. You can also key in a value in the **Depth** field and click to define the view direction. The value displayed in the **Depth** field is defined in model space, not paper space.
 The drawing view automatically appears on your cursor.

■ NOTES

- Selecting the **Update section** check box will cause an update of the related section view
 when the view is placed. This option is not checked by default, but the last setting is saved
 as a session file preference.
- The section view inherits the scale from the parent view by default.
 - TIP You can change the View Style or View Scale before placing the view.
- 1. Click to place the view on the sheet.

Edit a Cutting Plane

• To edit an existing cutting plane, select the cutting plane. The **Edit Cutting Plane** ribbon appears so you can edit the settings on the cutting plane. For more information, see *Edit Cutting Plane Ribbon* (on page 429).

Edit a Section View

 To edit an existing section view, select the section view. The Edit Section View ribbon appears so you can edit the settings on the section view. For more information, see Edit Section View Ribbon (on page 430).

Delete a Cutting Plane or Section View

- To delete a cutting plane, select it and press the DELETE key. If the cutting plane has an associated section view, a confirmation box appears allowing you to either convert the section view to a normal drawing view or delete the section view with the cutting plane.
- To delete a section view, select it and press the DELETE key. The associated cutting plane line remains, but only as a paper space graphic.

For more information on working with cutting planes, see *Place Cutting Plane/Section View Command* (on page 426).

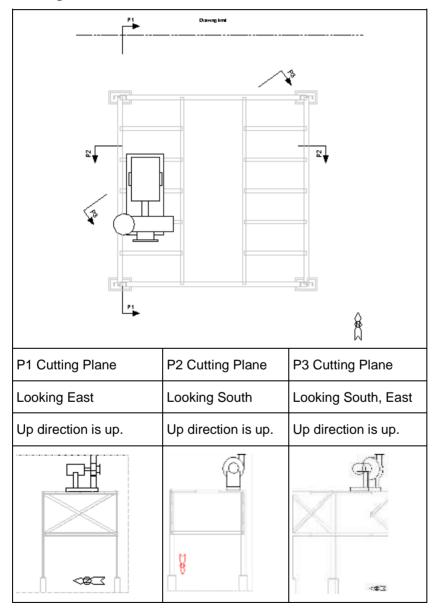
Section View Orientation Rules in Orthographic Drawings

Section View Orientation Behavior

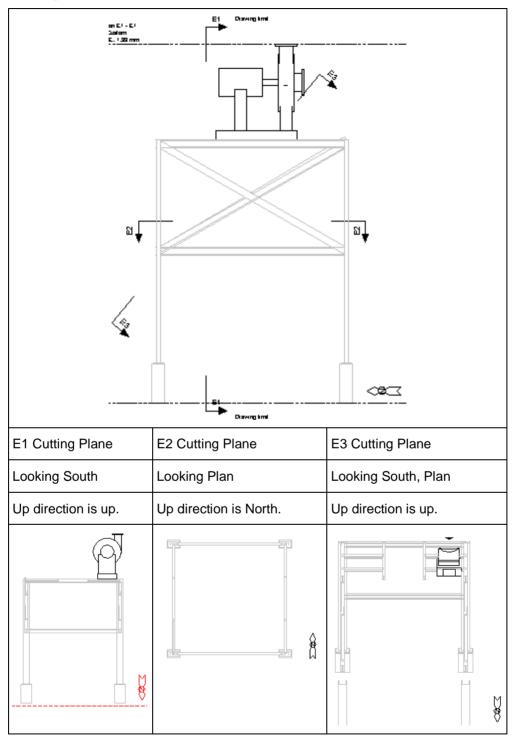
The software automatically determines the "up" direction for section views. In most cases, the up direction is "up," or +Z. In cases where using +Z is impossible (for example, if the section view direction is Looking Plan or Looking Up.), the software orients the view to the North direction.

The following examples outline common section view orientations:

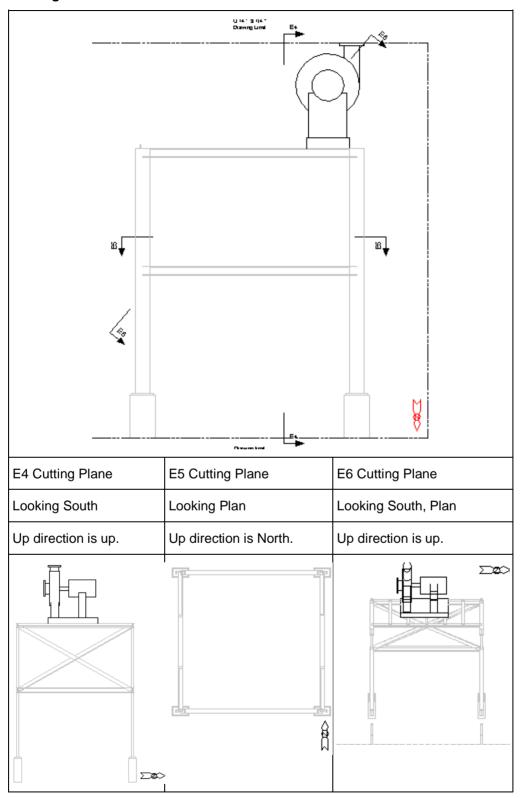
Looking Plan



Looking North



Looking East



Place a Section View

The following procedure describes editing a drawing document to add a section view.

Section views are extracted from main drawing views or other section views. Section views are similar to detail views, except that they display information in an orientation that is different from that of the originating view. The cutting plane is an annotation marker that indicates where to slice a needed group of objects and from which direction to look at that slice. The cutting plane is comprised of one or more line segments. When comprised of multiple line segments, cutting planes are referred to as "jogged" cutting planes.

■ NOTES

- For more information on the SmartSketch Drawing Editor commands used in this
 procedure, see Place Detail Envelope Command (on page 423) and Place Cutting
 Plane/Section View Command (on page 426) in the SmartSketch Drawing Editor Help.
- For information on the toolbar used in this procedure, see *Drawings Compose Toolbar* (on page 346) in the *SmartSketch Drawing Editor Help*.
- For information on the 2D commands available for editing, see the *SmartSketch Drawing Editor Help*.

Place a Cutting Plane

The following steps describe the basic workflow for placing a cutting plane on a drawing view used in creating a section view.

- 1. Open a drawing in **SmartSketch Drawing Editor**.
- 2. Click **Cutting Plane** \hookrightarrow on the toolbar.
- 3. In the drawing document, select a drawing view.
- 4. Type in text for the first arrow of the cutting plane in **Reference 1**. Type in text for the second arrow of the cutting plane in **Reference 2**.
- 5. To define the length of the cutting plane segment, click in the view to define the location of the first point, then click to define the location of the second point.
 - The cutting plane segment *displays* on the drawing, with arrows showing the default view direction.
- 6. To jog the cutting plane, click to define additional line segments. When you are ready to define the depth, right-click to proceed to the next step.
 - **NOTE** The cutting plane jogging behavior is not available in Ruleset drawings.
- 7. To change the view direction, move the cursor to the appropriate side of the cutting plane.

The direction automatically changes according to which side of the cutting plane segment the cursor is on.

NOTE The view direction is always perpendicular to the first cutting plane segment.

- 8. To define the section depth, click to define the distance and view direction. You can also key in a value in **Depth** and click to define the view direction.
 - **NOTE** The section depth is defined in model space, not paper space.

The drawing view automatically displays on your cursor.



- ▶ NOTE Selecting the **Update section** check box causes an update of the related section view when the view is placed. The **Update section** option is remembered globally for all section views.
- TIP You can change the View Style or View Scale before placing the view.
- 9. Click to place the view on the sheet.
 - **NOTE** The child view inherits the scale from the parent view by default.

Modify the Width, Depth, and Location of a Cutting Plane

The following steps describe the basic workflow for modifying the width of a cutting plane.

- 1. Open a drawing in SmartSketch Drawing Editor.
- 2. Click on an existing cutting plane.

The **Edit Cutting Plane** ribbon displays in the toolbar area.

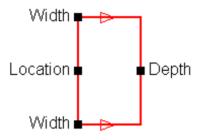
3. Click and drag the outside handles of the cutting plane segment in order to change its length.

■ NOTES

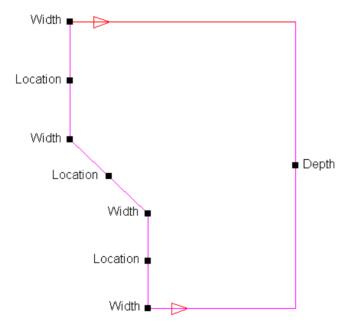
- The cutting plane handles correspond to the sides of the section view that are opposite to the section view depth handles. Changing the overall length of a cutting plane changes the corresponding sides of the related section view.
- In the case of a jogged cutting plane, each cutting plane segment will have its own modification handles.
- 4. To change the location of the cutting plane segment, click and drag the middle handle that lies between the width handles.
 - **NOTE** To change the location of an entire jogged cutting plane, click and drag the line segment; do not click a handle to move the jogged cutting plane.

5. Click and drag the handle opposite of the location handle to modify the cutting plane depth. You can also key in a value in **Section depth** and click in the view to define the view direction.

Single Cutting Plane



Jogged Cutting Plane



■ NOTE Selecting the **Update section** check box causes an update of the related section view when the **Finish** button is selected. The **Update section** option is remembered globally for all section views.

- 6. When the appropriate changes have been made to the cutting plane, select **Finish** in the **Edit Cutting Plane** ribbon.
- 7. In plant mode, right-click the view and select **Update**.

-OR-

In marine mode, right-click on the section view in the **Drawings View Explorer** and select **Update**.

Modify the Size of a Section View

The following steps describe the workflow for modifying the size of an existing section view.

- 1. Open a drawing in SmartSketch Drawing Editor.
- 2. Click on an existing section view.

The Edit Section View ribbon displays in the toolbar area.



NOTE The ribbon allows you to change the **View Style** and **View Scale**, as well as edit the **View Properties**.

- 3. Click and drag the handles on the top and bottom (or left and right sides) of the section view in order to change the view size.
 - ▶ NOTE Selecting the **Update section** check box causes an update of the section view when the **Finish** button is selected. The **Update section** option is remembered globally for all section views.
- When the appropriate changes have been made to the volume size, select Finish in the Edit Section View ribbon.
- 5. In plant mode, right-click the view and select **Update**.
 - -OR-

In marine mode, right-click on the section view in the **Drawings View Explorer** and select **Update**.

Place Detail View Command

Creates a detail view based on the selected detail envelope. A detail view is extracted from a main drawing view, a section view, or another detail view.

The drawing view style defines how objects appear in the detail views.

NOTE Detail views are based on detail envelopes. For more information, see *Place Detail Envelope Command* (on page 423).

Edit Detail View Properties

You can edit the properties associated with the detail view. When you access the properties through one of the following methods, the **Drawing View Properties** dialog box appears. For more information, see *Drawing View Properties Dialog Box (Drawing View Shortcut Menu)* (on page 467).

- Right-click the view and select **Properties** on the shortcut menu.
- Select the view, then select Edit > Properties.

Delete Behavior

To delete a detail view, select it and press **Delete**. Deleting the detail view does not delete the corresponding detail envelope.

See Also

Place Cutting Plane Ribbon (on page 428) Place a Detail View (on page 440) Place Detail View Ribbon (on page 440)

Place Detail View Ribbon

Sets options when placing or editing a detail view. This ribbon displays when you select a detail envelope, then click **Place Detail View**.

For information on placing detail envelopes, see *Place Detail Envelope Command* (on page 423).

Sheet Name

Indicates the drawing sheet where the detail view is placed. By default, the detail view is placed on the same sheet as the main drawing view.

View Style

Specifies a view style, which includes rules for filters, updates, and graphics. The view style controls the output characteristics of the view on the generated drawing. The list displays the 10 most recently used view styles in the session. Click **More...** to display the **Select View Style** dialog box.

NOTE Changing the view style for the detail view can result in symbology different from that used in the parent view.

View Scale

Sets the scale family and scale used for the detail view. The default scale setting is inherited from the parent view.

See Also

Place Detail View Command (on page 439)

Place a Detail View

The following procedure steps you through editing a drawing document to add a detail view.

A detail view is extracted from a main drawing view, a section view, or another detail view. Detail views are more than enlargements of the main drawing view. They often contain additional graphical information that is not visible in the main drawing view, such as weld or chalk information. You specify the detail view by drawing a circle or polygon around a portion of the main drawing view. The shape created is called the *detail envelope*.

■ NOTES

- For more information on the SmartSketch Drawing Editor command used in this procedure, see *Place Detail Envelope Command* (on page 423).
- For information on the toolbar used in this procedure, see *Drawings Compose Toolbar* (on page 346).
- For information on the 2D commands available for editing, see the *SmartSketch Drawing Editor Help*.

Place a Detail Envelope

The following steps describe the basic workflow for placing a detail envelope on a drawing to be used in creating a detail view.

NOTE Detail Envelope is only available when you create or open a Composed drawing from a 3D task.

- 1. Right-click a Composed drawing document in the **Detail View** and select **Edit**.
 - The drawing opens in SmartSketch Drawing Editor.
- 2. Click **Detail Envelope** on the toolbar.
 - The Place Detail Envelope ribbon bar displays.
- 3. In the drawing document, select a drawing view. Zoom in on the area of the main drawing view to the location you want to place the detail envelope geometry.
- 4. Click **Circle** ⊕ or **Polygon** △ so you can begin defining the detail envelope.
- 5. Type the text for the detail view name in **Reference mark**. If a second line of text is needed, type the text in **Additional callout text**.
- 6. For a **Circle** ⊕ detail envelope, click at the center of the circle, drag for the radius, and click again to set the radius.
- 7. For a **Polygon** riangle detail envelope, click points as needed to create the detail envelope geometry. Complete the polygon by moving the cursor over the starting point until the **Close Polygon** riangle symbol displays, then click to close the polygon.
- 8. You can now use this detail envelope to place a detail view in the drawing.

Place a Detail View

The following steps describe the basic workflow for placing a detail view based on a detail envelope.

- 1. Right-click a drawing document in the **Detail View** and select **Edit**.
 - The drawing opens in SmartSketch Drawing Editor.
- 2. In the drawing document, select a detail envelope to place a detail view. Select from the drawing area or in the Drawing View Explorer.
- 3. Click Place Detail View on the toolbar.
 - The Place Detail View ribbon bar displays.
- On the Place Detail View ribbon, make sure the settings are appropriate for your new detail view:
 - **Sheet Name** is disabled. By default, the view must be on the same sheet as the detail envelope.
 - Specify the View Style used for the view contents. Select More in the View Style list to view more style options.
 - Set the View Scale as needed. By default, the scale for the detail view is Fit to Scale.
- 5. Drag the preview outline of the view in the drawing area to the needed location. Click to place the view.
 - An outline of the view is placed. For marine mode, the view is also out-of-date in the Drawing View Explorer.
- 6. In plant mode, right-click the view in the drawing area and select **Update View**.
 - -OR-
 - In marine mode, right-click the view in the Drawing View Explorer and select Full Update.
 - Geometry, labels, and other annotations as defined by the view style display in the view.
- 7. Crop the view as needed by selecting the view and dragging the handles on the sides of the view.

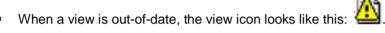
Delete a Detail Envelope from a Drawing

- 1. Right-click a drawing document in the **Detail View** and select **Edit**.
 - The drawing opens in SmartSketch Drawing Editor.
- 2. If a detail view has not been created, right-click a detail envelope in the Drawing View Explorer and select **Delete**.
- 3. If a detail view has been created, right-click the detail envelope and select **Delete**.
 - The Convert or Delete dialog box displays.
- 4. To convert the existing view(s) associated to the detail envelope to an independent view(s), select the Convert to independent drawing view(s) option and click OK. To delete the existing drawing view(s) associated to the detail envelope, select the Delete option and click OK.

Delete a Detail View from a Drawing (Marine mode only)

- 1. Right-click a drawing document in the **Detail View** and select **Edit**. The drawing opens in SmartSketch Drawing Editor.
- 2. In the Drawing View Explorer, right-click a view assigned to a drawing and select **Delete**.
- 3. Select **Delete** to delete the view permanently, or select **Unassign** to move the view in the Unassigned Folder.

- You can either move a view created by rules to the **Unassigned Folder** or permanently delete a view.
- If you delete a view that is a parent of other views, such as a detail or section view, the Convert or Delete dialog box displays. Select Convert to independent drawing view(s) to save the child view as an independent drawing view, or select **Delete** to delete the child view. The view is deleted.





- When a view is up-to-date, the view icon looks like this: 🔼
- When a view is unassigned and in the Unassigned Folder of Drawing View Explorer, the view icon looks like this:

2D/3D Selection Command

Allows you to move back and forth between the SmartSketch Drawing Editor application window and the 3D window. This command is only available in SmartSketch Drawing Editor when you create a new drawing or open an existing drawing from a 3D task. This command is turned off by default when you open a document in the SmartSketch Drawing Editor

Behavior while Turned On (Object Select/Highlight Mode)

When the command is on, you can select geometry in a drawing view, and the software highlights and selects the corresponding object in the 3D model. The object is selected even if it is not currently visible or displayed in the 3D workspace. If you select an object in the 3D task window, the software highlights the corresponding geometry in the drawing view in SmartSketch Drawing Editor. However, if the object is not included in the drawing view, no geometry highlights. The filters in the drawing view style dictate the content of the drawing view.

You can also multi-select 2D objects (groups and views) using the CTRL key.

NOTE If you select a drawing view (instead of geometry in the view), the 2D/3D Selection command turns off automatically and the Associate Objects to View 🍱 command is active. You can then modify the volumes or views associated to the selected view. For more information, see Associate Objects to View Command (on page 374).

Behavior while Turned Off (View Input Mode)

The command is off by default when you open a drawing.

When this command is off, geometry selected in a view is not actively linked to its 3D model object and the drawing document is in *view input* mode. You can modify the layout and contents of the drawing sheet and the properties of the drawing views.

▶ NOTE When 2D/3D Selection is off, you can click Associate Objects to View then move back and forth between the 3D application and SmartSketch Drawing Editor to associate drawing views to volumes in the model. For more information, see Associate Objects to View Command (on page 374).

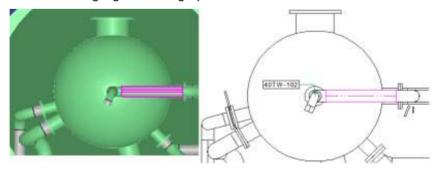
Considerations for Using the 2D/3D Selection Command

- 2D/3D Selection by default, only allows selection of 2D group elements that correspond to 3D model objects.
- You can use the spacebar key to toggle the selected drawing view when this command is used to associate objects to objects in the model. For more information, see Associate Objects to View Command (on page 374).
- Selecting a smart label in a SmartSketch Drawing Editor document selects the associated 3D object in the 3D application.
- Selecting a volume in the 3D application selects the associated views in the SmartSketch Drawing Editor document.
- The 2D select set is cleared when you close the drawing document.

Compare 2D Drawing Object to 3D Model Object

The following steps show you how to compare drawing objects to 3D model objects.

- 1. In a 3D modeling task, such as Common, select **Tools > Drawings Console**.
- 2. Right-click a drawing document and select **Edit** to open a drawing document in **SmartSketch Drawing Editor**.
- 3. In **SmartSketch Drawing Editor**, zoom into a drawing view to select the drawing object you want to compare to the 3D model.
- 4. Click **2D/3D Selection** to move focus to the 3D application. The 3D model object selects and highlights in the graphic windows.



Considerations for Using the 2D/3D Selection Command

The **2D/3D Selection** command, by default, only allows selection of 2D group elements that correspond to 3D model objects.

- You can use the spacebar key to toggle the selected drawing view when this command is used to associate objects to objects in the model.
- Selecting a SmartLabel in SmartSketch Drawing Editor document selects the associated 3D object in the 3D application.
- Selecting a volume in the 3D application selects the associated views in the SmartSketch Drawing Editor document.
- The 2D select set is cleared when you close the drawing document.

For more information on the commands available in **SmartSketch Drawing Editor**, see the *SmartSketch Drawing Editor Help*. You can also refer to the *Common User's Guide* for information on the **Tools > Drawings Console** command.

Using Scaled Sketching

In drawings, you can have multiple embedded views on a drawing sheet that are at different model scales (e.g. 1/4" = 1' and 1/8" = 1'). You can add graphics to the views and treat them as if they were actual model graphics. Use the Scaled Sketching command to draw all graphics at a consistent scale factor. The scale factor used is based on a user-selected SmartFrame.

For more information, see *Dimensioning Drawing Elements* in the *SmartSketch Drawing Editor User's Guide*.

See Also

Scaled Sketching Command (on page 446)
Scaled Sketching Ribbon (on page 447)
Draw or Edit Objects at the Same Scale Factor (on page 445)
Move a View with Scaled Sketching Objects (on page 447)

Draw or Edit Objects at the Same Scale Factor

Follow the steps below to use the Scaled Sketching command to draw or edit objects at a specific scale factor.

NOTE Make sure you have added the Scaled Sketching command to your Smart 3D toolbar. See Add the Scaled Sketching Button to the Smart 3D Toolbar.

1. Click Scaled Sketching to open the Scaled Sketching ribbon.



Scaled Sketching Ribbon (on page 447)

2. To set the color of any objects not being edited using the scale factor, click **Select color for referenced data** . Click on the required color from the color palette.



 Insert a SmartFrame (see Working with Object, Linking and Embedding) and set the scale factor of the SmartFrame, or click on an existing SmartFrame. The scale is read and taken from that SmartFrame. The selected scale type is displayed in the **Scaled Sketching** ribbon.

★ IMPORTANT

- Do not create or edit any layers when using the scaled sketching command.
- To get accurate dimensional data about scaled objects, be sure to dimension them
 while in scaled sketching mode. The dimensions are automatically configured to display
 at the value in which they were drawn.
- 4. Click **Enter Scaled Sketch mode M** to start drawing at the selected scale.

The system scales the drawing sheet to match the specified scale value.

- 5. Use the application's drawing/editing commands to draw the required graphics.
- 6. Click Finish to end the selected scale mode.

■ NOTES

- The only way to exit scaled sketching is to click Finish. All modifications you made in the document are retained.
- Whenever you finish using scaled sketching, the undo history is cleared and you cannot undo any previous edits. Undo is available after you continue working in SmartSketch Drawing Editor.
- Leaders must be placed on objects (Labels, Text Boxes, and so forth) that were placed during Scaled Sketch mode. If the objects were not placed during Scaled Sketch mode, the leader is not remembered in the SmartFrame after the view is updated.

Scaled Sketching Command

Enables you to edit or draw new objects at a scale that is different from the drawing scale for the active sheet. You can select a SmartFrame in the current document. The system then uses the scale factor of that SmartFrame.

See Also

Draw or Edit Objects at the Same Scale Factor (on page 445) Scaled Sketching Ribbon (on page 447)

Scaled Sketching Ribbon

The Scaled Sketching ribbon displays when you click **Scaled Sketching** on the Smart 3D **Drawings Compose** toolbar.

Select color for referenced data - displays a color palette for you to set the color of any objects not being edited using the scale factor.



Displays the scale taken from the selected SmartFrame.

Enter Scaled Sketch Mode M

Starts the system in using the scale factor for objects being drawn or edited.

Finish

Ends the scale mode.

See Also

Scaled Sketching Command (on page 446)

Draw or Edit Objects at the Same Scale Factor (on page 445)

Move a View with Scaled Sketching Objects

Follow the steps below to move a View that has objects drawn with the Scaled Sketching command.

- 1. If you are using the Scaled Sketching command, exit the command.
 - NOTE You cannot move a view while using the Scaled Sketching command.
- 2. Press ALT and click on the view you want to move.
- 3. Release ALT.
- 4. Move the view to a new location.

When you press and release **ALT** while clicking on the view, all of the Scaled Sketching objects move with the view.

Draw a Grate Opening with Scaled Sketching

Follow the steps below to use the **Scaled Sketching** command to draw a grate opening in a slab at a specific scale factor.

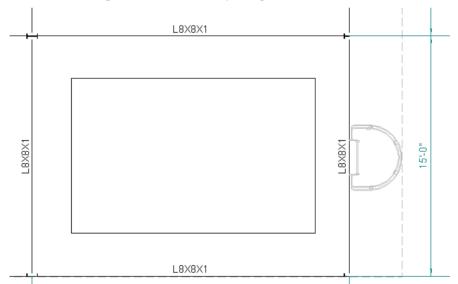
- 1. From the Drawings environment, open a drawing. In this example, the drawing contains equipment.
- 2. Click Scaled Sketching to open the Scaled Sketching ribbon.
- Click Select color for referenced data to set the color of any objects not being edited using the scale factor.
- 4. Click the SmartFrame in which you would like to sketch. The scale automatically matches the SmartFrame scale.

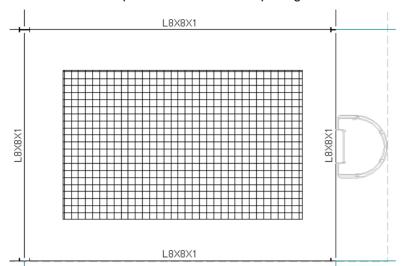
★ IMPORTANT

- Do not create or edit any layers when using the scaled sketching command.
- To get accurate dimensional data about scaled objects, be sure to dimension them
 while in scaled sketching mode. The dimensions are automatically configured to display
 at the value in which they were drawn.
- 5. Click **Enter Scaled Sketch mode a** to start drawing at the selected scale.

The system scales the drawing sheet to match the specified scale value.

6. Use the **Rectangle** tool to draw an opening on the slab.





7. Use the **Fill** tool to place a **Grid** fill in the opening.

8. Click **Finish** to end the selected scale mode.

■ NOTES

- Any objects drawn while in Scaled Sketching mode will move with the view if it is moved.
- The only way to exit scaled sketching is to click Finish. All modifications you made in the document are retained.
- Whenever you finish using scaled sketching, the undo history is cleared and you cannot undo any previous edits. **Undo** is available after you continue working in SmartSketch Drawing Editor.

Draw a Centerline with Scaled Sketching

Follow the steps below to use the Scaled Sketching command to draw a centerline on an equipment object at a specific scale factor.

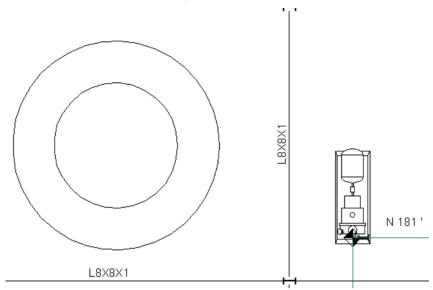
- 1. From the Drawings environment, open a drawing. In this example, the drawing contains equipment.
- 2. Click Scaled Sketching to open the Scaled Sketching ribbon.
- 3. Click **Select color for referenced data** to set the color of any objects not being edited using the scale factor.
- 4. Click the SmartFrame in which you would like to sketch. The scale automatically matches the SmartFrame scale.

★ IMPORTANT

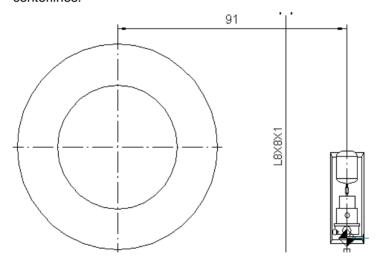
- Do not create or edit any layers when using the scaled sketching command.
- To get accurate dimensional data about scaled objects, be sure to dimension them
 while in scaled sketching mode. The dimensions are automatically configured to display
 at the value in which they were drawn.

5. Click **Enter Scaled Sketch mode a** to start drawing at the selected scale.

The system scales the drawing sheet to match the specified scale value.



- 6. Use the **Line** tool to draw a centerline on the equipment.
- 7. Use the **Distance Between** tool to place a dimension between the two equipment centerlines.



8. Click Finish to end the selected scale mode.

■ NOTES

- Any objects drawn while in Scaled Sketching mode will move with the view if it is moved.
- The only way to exit scaled sketching is to click Finish. All modifications you made in the document are retained.
- Whenever you finish using scaled sketching, the undo history is cleared and you cannot undo any previous edits. **Undo** is available after you continue working in SmartSketch Drawing Editor.

Draw a Textured Fill with Scaled Sketching

Follow the steps below to use the Scaled Sketching command to draw a grate opening in a slab at a specific scale factor.

- 1. From the Drawings environment, open a drawing. In this example, the drawing contains equipment.
- 2. Click Scaled Sketching to open the Scaled Sketching ribbon.
- 3. Click **Select color for referenced data** to set the color of any objects not being edited using the scale factor.
- 4. Click the SmartFrame in which you would like to sketch. The scale automatically matches the SmartFrame scale.

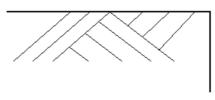
★ IMPORTANT

- Do not create or edit any layers when using the scaled sketching command.
- To get accurate dimensional data about scaled objects, be sure to dimension them while in scaled sketching mode. The dimensions are automatically configured to display at the value in which they were drawn.
- 5. Click Enter Scaled Sketch mode \(\mathbb{Z} \) to start drawing at the selected scale.

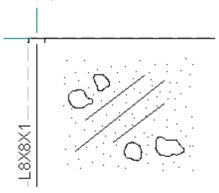
The system scales the drawing sheet to match the specified scale value.

6. Use the Fill tool to place an Earth fill texture inside a shape.

Earth



TIP You can also draw your own fill using **Scaled Sketching**. Below is an example of a manually drawn concrete fill.



7. Click Finish to end the selected scale mode.

■ NOTES

- Any objects drawn while in Scaled Sketching mode will move with the view if it is moved.
- The only way to exit scaled sketching is to click Finish. All modifications you made in the document are retained.
- Whenever you finish using scaled sketching, the undo history is cleared and you cannot undo any previous edits. **Undo** is available after you continue working in SmartSketch Drawing Editor.
- Fills can be placed while in scaled sketching mode if they are bounded by view objects, scaled sketch objects, or a combination of both.
- Fills are automatically deleted when you click Finish on the Scaled Sketching toolbar and:
 - Use scaled sketching for view A and place a fill on a scaled sketch object in view B.
 - Use scaled sketching for view A and place a fill on an object in view B.
 - Use scaled sketching for a view and place a fill on a non-scaled sketch, manually-drawn graphic.

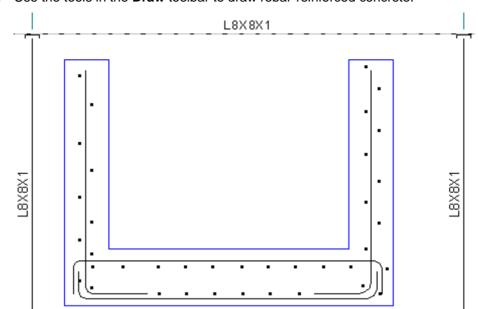
Draw Rebar with Scaled Sketching

- 1. From the Drawings environment, open a drawing. In this example, the drawing contains equipment.
- 2. Click Scaled Sketching to open the Scaled Sketching ribbon.
- 3. Click **Select color for referenced data** to set the color of any objects not being edited using the scale factor.
- 4. Click the SmartFrame in which you would like to sketch. The scale automatically matches the SmartFrame scale.

★ IMPORTANT

- Do not create or edit any layers when using the scaled sketching command.
- To get accurate dimensional data about scaled objects, be sure to dimension them
 while in scaled sketching mode. The dimensions are automatically configured to display
 at the value in which they were drawn.
- 5. Click **Enter Scaled Sketch mode a** to start drawing at the selected scale.

The system scales the drawing sheet to match the specified scale value.



Use the tools in the **Draw** toolbar to draw rebar-reinforced concrete.

7. Click Finish to end the selected scale mode.

■ NOTES

- Any objects drawn while in Scaled Sketching mode will move with the view if it is moved.
- The only way to exit scaled sketching is to click Finish. All modifications you made in the document are retained.
- Whenever you finish using scaled sketching, the undo history is cleared and you cannot undo any previous edits. **Undo** is available after you continue working in SmartSketch Drawing Editor.

Draw an Opening in a Plate with Scaled Sketching

Follow the steps below to use the **Scaled Sketching** command to draw an opening in a plate at a specific scale factor.

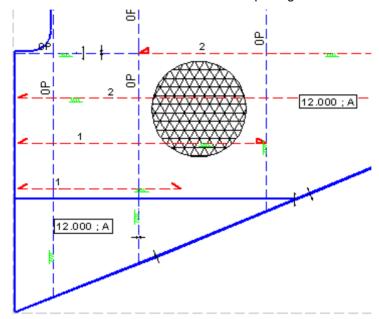
- 1. From the Drawings environment, open a drawing. In this example, the drawing contains equipment.
- 2. Click Scaled Sketching to open the Scaled Sketching ribbon.
- 3. Click **Select color for referenced data** to set the color of any objects not being edited using the scale factor.
- 4. Click the SmartFrame in which you would like to sketch. The scale automatically matches the SmartFrame scale.

★ IMPORTANT

- Do not create or edit any layers when using the scaled sketching command.
- To get accurate dimensional data about scaled objects, be sure to dimension them
 while in scaled sketching mode. The dimensions are automatically configured to display
 at the value in which they were drawn.

- 5. Click Enter Scaled Sketch mode at to start drawing at the selected scale.

 The system scales the drawing sheet to match the specified scale value.
- 6. Use the **Circle** tool to draw an opening on the plate.
- 7. Use the Fill tool to add a Mesh fill to the opening.



8. Click **Finish** to end the selected scale mode.

■ NOTES

- For information on how to move a view that contains objects drawn using the Scaled Sketching command, see Move a View with Scaled Sketching Objects (on page 447).
- If a view is deleted and moved to the UnAssigned Folder, any sketches made in Scaled Sketching mode reside in the view. If the view is reassigned to a drawing, your sketches will appear in the view.
- The only way to exit scaled sketching is to click Finish. All modifications you made in the document are retained.
- Whenever you finish using scaled sketching, the undo history is cleared and you cannot undo any previous edits. **Undo** is available after you continue working in SmartSketch Drawing Editor.

See Also

Using Scaled Sketching (on page 445)

Retain Edits made inside a Drawing View

- 1. Double-click a drawing view's border to open the drawing view.
- 2. Make the required changes to the drawing view.
- 3. Click **File > Update** to save the changes.
- 4. Click File > Close.

- 5. Repeat the steps above as required for all other views.
- 6. Save and close the document.

The changes that are saved include:

- Layer-related changes (creating new layers, modifying a graphic's layer, and changing how layers are displayed using the Display Manager Dialog Box).
- Using the Drawing Editor to sketch new graphics, copy intelligent graphics, add text boxes, place fills, or place symbols.

When a drawing view is opened, the new window is maximized to fit the graphics within the view. Like the Scaled Sketching command, the sheet scale changes to match the view scale. If the view scale is not Fit to Scale, manual graphics drawn outside of the rectangle, enclosing the graphics within the view, are not visible after the next update.

■ NOTES

- Particular changes made to intelligent graphics affects what is retained when a drawing view is updated:
 - Copies of intelligent graphics are treated as manual graphics.
 - When no graphic rule is specified for a filter, or when the graphic rule does not specify a layer, intelligent graphics are placed onto the **Default** layer.
 - Intelligent graphics remain intelligent graphics after their layer is changed. If the corresponding 3D object is deleted, the layer change operation is lost.
 - It is recommended that you include any dimensions or leaders in the drawing sheet, instead of the drawing view, as dimensions and leaders connected to intelligent graphics are not retained when the view is updated.
 - Intelligent graphics that are deleted will reappear after the drawing view is updated.
 Select **Hide** to temporarily remove intelligent graphics.

Hide/Show Object Command

Hides or displays objects in the drawing view. You can hide certain objects from the drawing view by selecting object types from a list. You can also show objects that you have hidden using the same command.

When objects are hidden, they are added to a different drawing layer that is created automatically. When the objects are shown, they are restored to their original drawing layer and all annotations are restored.

■ NOTE You can also hide/show symbol objects that have Replace Object(s) with Symbol rule applied on them.

See Also

Hide/Show Object Ribbon (on page 456) Hide an Object (on page 457) Show an Object (on page 457) Copy an Object (on page 458)



Subtracts objects from the view.



Adds objects to the view.

View Name

Displays the selected view name.

Hide Options

Displays the list of hide options available in the selected drawing view. The default option is AII.

- All Hides graphics, labels and dimensions.
- Graphics only Hides only graphics.
- **Graphic/Labels** Hides graphics and corresponding labels.
- **Graphic/Dimensions** Hides graphics and related dimensions.

Object Filter

Displays a list of object types in the selected drawing view. The default option is All. When you select an object in the drawing, the Object Filter displays associated object type automatically.



Copy Graphics

Copies only graphics into the smart frame. Ignores any applied hide options. By default it is disabled. Click **Hide** to enable this command.

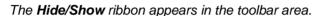
Finish

Saves and exits the Hide/Show command. Click on the drawing view to enable this command.

NOTE You must update the drawing after hiding or showing objects.

Hide an Object

1. In the **Draw** toolbar, click **Hide/Show** ...



NOTE The Hide mode is selected by default.

2. Select a drawing view.

Hide gets disabled.

- 3. Select an hide option from the Hide Options list.
- 4. Select an object type from the **Object Filter** list.
- 5. Select the objects you want to hide.

As you select the objects, the Finish command is enabled, and the selected objects are highlighted in the drawing view depending on the hide option. For example, if the hide option is Graphics/Labels, the selected graphic and related labels are highlighted in the drawing view.

6. Click **Finish** to save your changes.

NOTE You must update the drawing after hiding objects.

Show an Object

1. In the **Draw** toolbar, click **Hide/Show** ...

The Hide/Show ribbon appears in the toolbar area.

- 2. Click Show \$\frac{\x}{\square}\$.
- 3. Select a drawing view.

All hidden objects in the selected view are highlighted. Except for **Show** , the remaining options on the ribbon bar are disabled and set to their respective default values.

4. Select the objects you want to hide. Only highlighted objects are displayed in the drawing view

The Finish command is enabled.

Click Finish to save your changes.

NOTE You must update the drawing after showing objects.

Copy an object

1. In the **Draw** toolbar, click **Hide/Show**



NOTE The Hide amode is selected by default.

- 2. Select a drawing view.
- 3. Click Copy Graphics .
- 4. Select an object to copy.
- 5. Click **Finish** to save your changes.

You can see the copied object with no labels.

NOTE To view the copied object, double-click the drawing view. Move the copied object, and then turn on the hidden layer. For more information on layer display, see *Layers Command* in the *SmartSketch Drawing Editor User's Guide*.

Copy and Paste View Command

Copies an orthographic drawing view and places the copy on the same sheet.

- 1. Select an orthographic drawing view, and then click **Copy and Paste View \(\begin{array}{c}\) = \(\begin{array}{c}\)**
- 2. On the target drawing sheet, click **Paste 8**.

The view displays.

- 3. Drag the drawing view to a new location on the sheet.
- TIPS You must use Copy and Paste View a rather than the traditional Cut, Copy, and Paste commands on views if you want to maintain the associations.
- If you try to copy a view, the software displays a message telling you that the copied view will be an unassociated view. You then have the choice to continue or not.

Move View Command

Moves one or more views from a composed drawing document to another composed drawing document. The new drawing can be in the same component or in a different component.

A moved section view keeps its association with its cutting plane in the parent view, even when the parent view is in a different drawing. A moved detail view keeps its association with its envelope in the parent view, even when the parent view is in a different drawing.

Report and key plan views are moved automatically if the parent view is also moved. Dependent report and key plan views cannot be moved if the parent view is not moved.

Views cannot be moved to a drawing that is being edited or to a drawing to which you do not have write permissions.

■ NOTES

- Views cannot be moved in orthographic drawings by query or volume (spatial) drawings.
- For a marine mode drawing by rule, views cannot be moved to a document in a different component with **Move View *****. Views can be moved to a document in the same component, and to a different sheet in the same document.

Move View Dialog Box (on page 460)

What do you want to do?

- Move a view to a different drawing (on page 459)
- Move multiple views to a different drawing (on page 459)

Move a view to a different drawing

- 1. Click Move View 34.
- 2. Select a view in the drawing area.
 - **NOTE** Alternatively, select the view first, and then click **Move View 34**.

The Move View dialog box appears.

- 3. Expand the appropriate folder and component, and select the destination drawing document.
- 4. Click OK.

The selected view is moved to the destination drawing.

Move multiple views to a different drawing

- 1. Select two or more views in the drawing area.
- 2. Click Move View 34.

The **Move View** dialog box appears.

- Expand the appropriate folder and component, and select the destination drawing document.
- 4. Click OK.

The selected views are moved to the destination drawing.

Move View Dialog Box

Displays the Drawings and Reports tree view of folders, components, and drawings in the model. The tree view displays only drawings to which you have write permissions. Expand the appropriate folder and component, and then select the drawing to which you want to move your selected views.

OK

Closes the dialog box and moves the view to the selected drawing document.

Cancel

Closes the dialog box without moving the view.

NOTE For drawings by rule, you can only move the selected view to drawings and drawing sheets under the same component as the selected view. Only the drawings and sheets in that component are displayed.

Highlight Annotations Command

Highlights labels, dimensions, and customized graphics based on the options that you select. The options display on the left side of the drawing window.

This command is useful when you are troubleshooting labels and dimensions, such as when you update a drawing and are looking for certain label or dimension types.

Highlight Dialog Box (on page 460)

Highlight Dialog Box

Allows you to specify label and dimensions that you want to see highlighted on a drawing. This dialog box appears docked on the left side of the drawing by default. You can undock it if needed.

To see all of the options, use the scroll bar on the right side of the dialog box.

Highlight

Click this button to highlight the specified items in the drawing.

Clear

Clears all highlighted items on the drawing and removes all the items in the select set.

Clear Options

Clears all options on this dialog box.

Close

Ends this command.

Add to Select Set

Adds the highlighted items to the set of all selected items.

Choose Highlight Color



Specifies the color of the selected items on the drawing.

Labels

Unmodified System Placed

Highlights labels that have not been modified on the drawing and that were placed by the software.

User Placed

Highlights labels that a user placed on the drawing.

Modified System Placed

Highlights labels that have been modified on the drawing and that were placed by the software.

Deleted

Highlights labels that have been deleted.

Unassociated

Highlights manual labels that are no longer associated to at least one of their original geometry elements. Loss of association can occur after a view update when the **Default Graphics** or **Drawable Default Graphics** custom graphic modules are used in a view style. For more information, see *Default Graphics* and *Drawable Default Graphics* in the *Drawings and Reports Reference Data Guide*.

Additional Label Filters

Labels By Name

Click this option in order to choose a specific label type. When you check this box, the dropdown list is enabled. You can choose **More** in the dropdown list to view all label types in the current drawing.

Corrupt

Highlights labels that have a problem, such as not being connected to the correct object.

Include

Include Related Leaders

Highlights leaders that are related to the specified labels.

Include Other Relations

Highlights cutting planes or detail envelopes that are related to the specified labels.

Include Related Lines

Highlights lines that are related to the specified labels.

Dimensions

Unmodified System Placed

Highlights dimensions that have not been modified and that were placed by the software.

User Placed

Highlights dimensions that a user placed on the drawing.

Modified System Placed

Highlights dimensions that have been modified and that were placed by the software.

Deleted

Highlights dimensions that have been deleted.

Paper To Model

Highlights dimensions from an object drawn on the paper to an object that is in the model.

Unassociated

Highlights manual dimensions that are no longer associated to at least one of their original geometry elements. Loss of association can occur after a view update when the **Default Graphics** or **Drawable Default Graphics** custom graphic modules are used in a view style. For more information, see *Default Graphics* and *Drawable Default Graphics* in the *Drawings and Reports Reference Data Guide*.

Additional Dimension Filters

Dimensions By Name

Click this option in order to choose a specific dimension type. When you check this box, the dropdown list is enabled. You can choose **More** in the dropdown list to view all dimension types in the current drawing.

User Graphics

User Graphics

Highlights items, such as lines, that a user placed on the drawing.

User Scaled Dimensions

Highlights dimensions on the items that a user placed on the drawing.

Leaders

Disconnected Leaders

Highlights leaders that are disconnected from labels on the drawing.

See Also

Highlight Annotations Command (on page 460)

Clear Manual Edits Command

Permanently clears all manual edits made to labels and dimensions in the selected views.

You must first select one or more views before clicking this command.

Associate Graphics to Graphic View Command

Associates and disassociates manually drawn objects, such as lines, circles, symbols, and text boxes, to graphic views. You can use Associate Graphics to Graphic View 📓 in composed drawings and Drawings by Rule components. You cannot associate objects with a report view or a pasted view, or if the objects are already associated with another view. To associate the objects to another view, you must first dissociate the objects.

■ NOTES

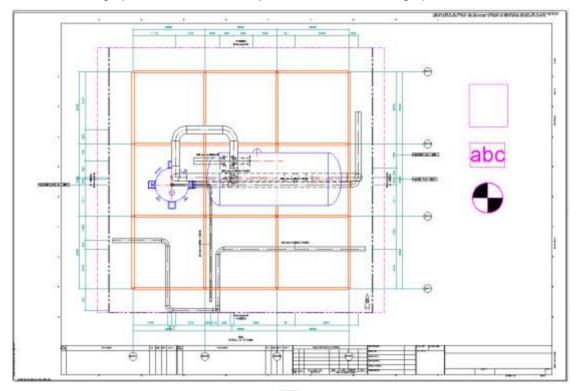
- You can use **Move View** it to move the view and the associated objects.
- To view all objects that are associated with a graphic view, press ALT and select the graphic view.

What do you want to do?

- Associate Graphics to Graphic View (on page 463)
- Disassociate Graphics from Graphic View (on page 465)

Associate Graphics to Graphic View

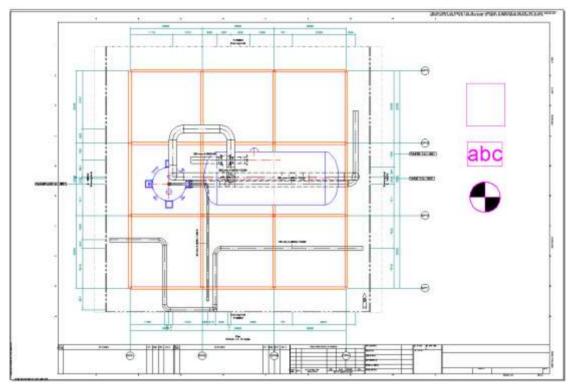
- 1. Select a graphic view.
- 2. In the selected graphic view, select the objects to associate to the graphic view.



- 3. Click **Associate Graphics to Graphic View** ... *A message displays the number of associated objects.*
- 4. Click OK.

Disassociate Graphics from Graphic View

1. Select the objects to disassociate from the graphic view.



- 2. Click Associate Graphics to Graphic Views
 - A warning message displays.
- 3. Click Yes to remove all associated inputs from each object.
 - A message displays the number of disassociated objects.
- 4. Click OK.

Template Toolbar

This toolbar is available when you click **Edit Template** on a component shortcut menu.

Place Drawing View Command (Template Toolbar)

Places a two-dimensional drawing view for 3D volume drawings. This command is available when you use the **Edit Template** command available on the shortcut menu for a volume drawing component in the Drawings and Reports task.

When you place drawing views, the software automatically saves the views to the **DwgTemplate** layer when you save the document.

See Also

Manual Place View Ribbon (on page 469)



The drawing view highlights on the drawing and the **Drawing View Properties** dialog box displays.

- 5. On the **Drawing View Properties** dialog box, define the necessary information on the **View** tab in the **Name** and **Description** boxes.
- 6. In the **Style** box, select a view style.
- 7. In the **Orientation** box, select an orientation such as **Looking Plan**.
- 8. In the **Scale** box, select a scale, or choose **Custom** and define the values in the boxes to the right.

TIPS

- If you choose Custom, you must type values that are greater than zero in the boxes at the right.
- The software converts the values that you type to the default unit of measure. For example, if you type values of 1 in to 1 ft, the software may convert those values to mm.
- 9. Specify additional information as necessary on the **Format** tab.
- 10. Click **OK**.
- 11. Click File > Save and exit SmartSketch Drawing Editor.

■ NOTES

- To associate volumes with the view, switch to the Space Management task and click one of the Place Drawing Volume commands.
- You can resize a drawing view by dragging its handles.
- You cannot undo a drawing view delete operation. A message box displays when you press
 Delete for a selected view, providing a chance to cancel the operation.

Drawing View Properties Dialog Box (Drawing View Shortcut Menu)

Sets drawing view properties for a 3D volume drawing. This dialog box appears when you select a drawing view in a volume drawing then right-click and select **Properties** on the shortcut menu.

See Also

Place Drawing View Command (Template Toolbar) (on page 466) Info Tab (Drawing View Properties Dialog Box) (on page 467) Format Tab (Drawing View Properties Dialog Box) (on page 467) View Tab (Drawing View Properties Dialog Box) (on page 467)

Info Tab (Drawing View Properties Dialog Box)

Provides information about the frame around a drawing view. This information is view-only. You cannot make changes.

Type

Displays the category of the selected element.

Sheet

Displays the name of the drawing sheet that contains the selected element.

Layer

Shows the layer that contains the selected element.

Origin

Specifies the coordinates, or location, of an element along the X- and Y-axes.

See Also

Place Drawing View Command (Template Toolbar) (on page 466)
Place Drawing View Command (Template Toolbar) (on page 466)
Order Drawings (on page 354) for information on the properties NOTES

- For Custom scale, the default is to the document's unit of measure setting. For example, if you define values of 1 in to 1 ft, the values are converted to mm if that is the default unit of measure.
- Do not use negative values when defining custom scale values.
- For volume drawings, if the volume is too big for the drawing view, the software centers the volume in the drawing view.

Navigation rule

Specifies the navigation rule to use to traverse elements to be included in the range for Orthographic Drawings by Query. This dropdown only appears when you are placing views on drawings created by the Orthographic Drawings by Query component in the Drawings and Reports 3D task. The navigation rule can also optionally return separate object collections, whose range is included in the 3D object range. If no navigation rule is specified, the drawing object collection includes everything in the 3D object range. The delivered rules are:

- HngSupSimpleNavigator.dll Specific to Hanger and Support objects. Returns the HangersSupport objects, support components, supporting objects, supported objects, and the design children (recursively). It also returns the control points on the HangerSupport objects and support components.
- HngSupRangeNavigator.dll Same as HngSupSimpleNavigator.dll with design children collection to extent the HangerSupport range.
- DrawingSpoolNavigator.dll Specific to Spools. Returns the Spool, its connected parts, and their features.
- AssemblyNavigator.dll Specific to Assemblies. Returns the assemblies, pipe spool, penetration spool, its connected parts, and their features.

Convert report output to text boxes (no Excel)

Specifies that any report associated with this drawing view will be converted automatically to native text box format, even if the report is an Excel spreadsheet report. For information on converting Excel spreadsheet reports, see *Convert Excel Spreadsheet Reports to Native Text Box Format Custom Command* (on page 491).

Flush Threshold

Sets a parameter for memory management. When the number of objects processed for a drawing document reaches the **Threshold** value, they are removed from memory. If they are needed later, they are recalled from the database. This property helps improve drawing update performance. The **Flush Threshold** property is only available for composed drawing documents. The default value is **2000**, with a range of **5-5000**. The higher number is faster but uses more memory, which is acceptable for smaller drawings. Lower numbers are slower but allow larger drawings to complete faster.

NOTE If a drawing document does not successfully update in the Drawings and Reports 3D task, check the error log for the drawing document. If the error log shows memory overflow errors, lower the **Flush Threshold** value.

VHL Precision

Sets a parameter for Hidden Line removal processing to consider two lines as identical. This property setting has a direct impact on the visibility of the lines in the drawing. It improves drawing update performance, but it may reduce drawing quality. The **VHL precision** property is only available for composed drawing documents. The default value is **0.000001**, with a range of **0.001 to 0.000001**. The smaller the number, the more accurate the graphic elements are in the 2D view.

NOTE If some intersections of complex surfaces appear on/off along the curve, the precision of the VHL may be too restrictive compared to the precision by which the surfaces were created. Lowering the **VHL Precision** value may help the display of the intersections, but lowering the value too much could degrade the overall quality and the visibility of the drawing details.

Geometry Validation

Sets a parameter for drawing completion and quality to improve drawing update performance. The **Geometry Validation** property is available for composed drawing documents only. The default value is **Off**. When set to **Off**, the drawing document completes, but invalid geometries are left out. If set to **On**, the drawing document does not complete if invalid geometries are encountered during update.

Tag

Identifies the reference mark for the selected detail view. This property is only available when a detail or section view is selected.

Caption

Identifies the callout text assigned to the selected detail view. This property is only available when a detail or section view is selected.

See Also

Place Drawing View Command (Template Toolbar) (on page 466) Place Drawing View Command (Template Toolbar) (on page 466)

Place Report Command (Template Toolbar)

Embeds a report in a drawing view on a 3D drawing. You must select a drawing view on the drawing before you can embed a report.

The report queries on the items in the drawing view.

When you place reports, the software automatically places them on the **DwgTemplate** layer so that they will be preserved when you update the drawing document.

■ NOTES

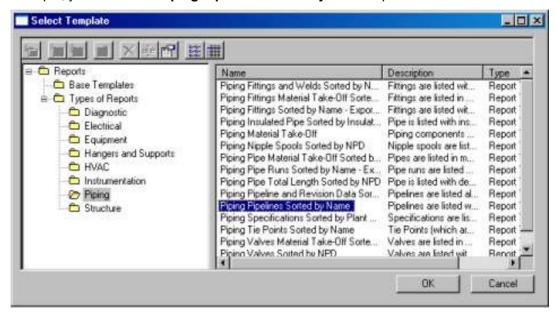
- For embedded reports to run on computers with Microsoft Office XP, you must modify the security settings in Excel to allow Visual Basic projects to run. To change this setting, open Excel, and click Tools > Macros > Security. On the Trusted Sources tab, select Trust access to Visual Basic Project. This setting must be modified before you update the drawing and generate the report within it.
- When defining embedded report layout (sizing of columns and rows), consider the report usage first. Because of a Microsoft limitation concerning the size of Windows metafile objects within other applications, the data displayed may be incomplete. Therefore, no column should be out of screen when using 100 percent zoom for the report. Otherwise some columns are ignored when the report is embedded within the drawing. The same limitation exists for rows. To preserve the maximum number of rows displayed, the total header row(s) height should be a minimum of the overall report. Using Microsoft Excel default settings, the maximum number of columns is approximately 20 and the maximum number of rows is approximately 75 (including header rows). For more information on setting the defaults in Microsoft Excel, see your Microsoft Excel documentation.

See Also

Report Properties Dialog Box (Place Report View Command) (on page

Place an Embedded Report in a Volume Drawing

- 1. For volume drawings, right-click the volume drawing component in the **Drawing Console** and select **Edit Template**.
- 2. Select a template on the **Select Template** dialog box and click **OK**. The drawing template opens in **SmartSketch Drawing Editor**.
- 3. Click **Place Report** , and then click on the drawing view border.
- 4. On the **Select Template** dialog box, select a report template from the hierarchy. For example, you can select **Piping Pipelines Sorted by Name** report.



- 5. Click **OK** on the dialog box.
- 6. In SmartSketch Drawing Editor, click to place the report.

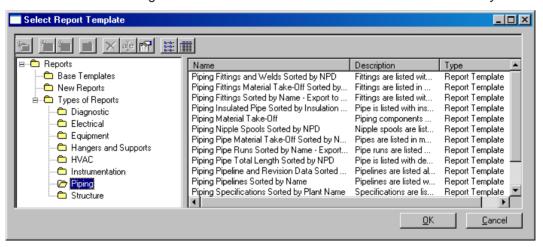
■ NOTES

- Only one report per drawing view is permitted.
- For more information about Microsoft Office and service packs, refer to the *Microsoft web* site (http://www.microsoft.com/) (http://www.microsoft.com/).
- If you are using Office 2003, in Microsoft Excel under Tools > Macro > Security > Trusted
 Publishers tab, check the Trust Access to Visual Basic Project option.
- If you are using Office 2007 and Office 2010, click the Microsoft Office button to access Excel Option. Go to the Trust Center category and select the Trust Center Settings button. Select the Macro Settings category and check Trust access to the VBA project object model.

When defining embedded report layout (sizing of columns and rows), consider the report usage first. Because of a Microsoft limitation concerning the size of Windows metafile objects within other applications, the data displayed may be incomplete. Therefore, no column should be out of screen when using 100 percent zoom for the report. Otherwise some columns are ignored when the report is embedded within the drawing. The same limitation exists for rows. To preserve the maximum number of rows displayed, the total header row(s) height should be a minimum of the overall report. Using Microsoft Excel default settings, the maximum number of columns is approximately 20 and the maximum number of rows is approximately 75 (including header rows). For more information on setting the defaults in Microsoft Excel, see your Microsoft Excel documentation.

Select Report Template Dialog Box

Selects a report template. This dialog box appears when you click **Place Report** when editing a 3D drawing template. By browsing through the hierarchy, you can find any report template in the Catalog database. After you select a template, the software generates the report. You can resize the dialog box and the columns to view the information more clearly.



Properties

Displays the properties of the selected item. All properties on the **Properties** dialog box are read-only.

List View

Sets the dialog box to display items in a list view.

Grid View

Sets the dialog box to display items in a spreadsheet-style grid view.

NOTE The **Place Report** commands creates reports based on a selected report template. You can also select a report template and view its properties. The buttons that are grayed out are not available when using these commands.

Place Key Plan Command (Template Toolbar)

Places a key plan on a drawing template. The Place Key Plan command is enabled when you select a drawing view. The command displays the **Select Key Plan** dialog box to specify a key plan type to associate with the selected drawing view.

When you place key plan views, the software automatically places them on the **DwgTemplate** layer so that they will be preserved when you update the drawing document.

A key plan is a small graphical representation of a drawing volume you defined in the geographic area where engineering is taking place. One key plan may serve for an entire project. Multiple key plans may be established per discipline. One drawing view may have multiple key plans.

See Also

Select Key Plan Style Dialog Box (on page 472)

Select Key Plan Style Dialog Box

Specifies a style for the selected key plan on a volume- drawing template. Select a style from the **Rule Name** hierarchy.

See Also

Key Plan Properties Dialog Box (on page 472) Place Key Plan Command (Template Toolbar) (on page 472)

Key Plan Properties Dialog Box

Sets options for a key plan on a drawing template.

See Also

Place Key Plan Command (Template Toolbar) (on page 472) Info Tab (Key Plan Properties Dialog Box) (on page 472) Format Tab (Key Plan Properties Dialog Box) (on page 473) Key Plan Tab (Key Plan Properties Dialog Box) (on page 473)

Info Tab (Key Plan Properties Dialog Box)

Provides information about a key plan. This information is read-only. You cannot make changes.

Type

Displays the category of the selected element.

Sheet

Displays the name of the drawing sheet that contains the selected element.

Layer

Shows the layer that contains the selected element.

Origin

Specifies the coordinates, or location, of an element along the X- and Y-axes.

Format Tab (Key Plan Properties Dialog Box)

Formats the frame around a key plan.

Show Border

Displays the frame around the object.

Color

Sets the color of the frame.

Line Width

Sets the line thickness on the frame.

Line Type

Overrides a line type for a drawing sheet or embedded object and sets another line style for an element or a linked object.

See Also

Place Key Plan Command (Template Toolbar) (on page 472) Key Plan Properties Dialog Box (on page 472)

Key Plan Tab (Key Plan Properties Dialog Box)

Sets the properties for a selected key plan.

Name

Specifies a unique name for the key plan.

Description

Describes the key plan contents.

Style

Indicates the key plan view style used. Select **More** to display the **Select Key Plan Style** dialog box. For more information, see *Select Key Plan Style Dialog Box* (on page 472).

Scale

Indicates the scale assigned to the key plan with regard to the associated drawing view. If you select **Custom** for the scale, you must type values that are greater than zero in the boxes at the right to set the scale-to-scale ratio for key plan-to-drawing view.

■ NOTES

- For Custom scale, the default is to the document's unit of measure setting. For example, if you define values of 1 in to 1 ft, the values are converted to mm if that is the default unit of measure.
- Do not use negative values when defining custom scale values.
- Orientation of the key plan graphic is specified as part of the key plan style.

See Also

Key Plan Properties Dialog Box (on page 472) Place Key Plan Command (Template Toolbar) (on page 472)

Place a Key Plan

- In the Management Console, right-click a volume drawings component, then click Edit Template.
- 2. If no template is defined for the volume drawings component, select a template in the **Select Template** dialog box, and click **OK**.
- 3. Select a drawing view on the template.
- 4. On the **Template** toolbar, click **Place Key Plan**
- 5. Click two points to place the key plan view.



On the second click, the software displays the **Key Plan Properties** dialog box.

6. On the dialog box, specify the settings for the key plan.

■ NOTES

- Multiple key plans can be associated with a single drawing view.
- For Custom scale, the default is to the document's unit of measure setting. For example, if you type values of 1 in to 1 ft, the values are converted to mm if that is the default unit of measure.
- Do not use negative values when typing custom scale values.
- Orientation of the key plan graphic is specified as part of the key plan style. For more information, see "Define View Style Command (Tools Menu)" in the *Drawings and Reports* Reference Data Guide.

For more information, see *Key Plan Drawing Requirements* and *Key Plan View Styles* in the *Drawings and Reports Reference Data Guide.*

Edit Border Template Toolbar

Available when you click **Tools > Edit Border Template** in the Drawings and Reports task.

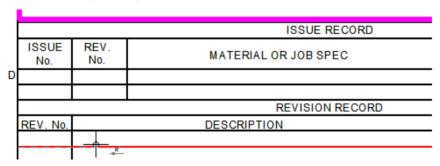
Place Drawing Property Label Command (Drawing Labels Toolbar)

Positions drawing property labels in the title block of a template. The command allows you to place drawing properties defined in the 3D drawing XML schema file as title block information

on the drawing. When you click **Place Drawing Property Label**, the **Place Drawing Property Label** ribbon appears in the toolbar area.

Place Drawing Property Label Ribbon (on page 475)

When placing drawing property labels, zoom into the border area where you want to place the label for more precise placement.



Place Drawing Property Label Ribbon

Sets options for drawing property label placement on a border template. You can access this ribbon when you click the **Place Drawing Property Label** command when editing a border template in SmartSketch Drawing Editor.

When you place drawing property labels, the software automatically makes the **DwgTemplate** layer active. The labels need to be on this layer so that they are preserved when you update the drawing.



You set options as needed within the **Place Drawing Property Label** ribbon, then place the label in the drawing border title block area.

Label Set

Specifies a property category. This list shows the categories of drawing properties available for the current drawing. The label set controls the fields listed in the Field dropdown list and the enabling of other options on the ribbon.

Fields

Lists the properties available in the selected Label Set. This is the information you are placing on the title block of the drawing.

Function

Provides positioning functions for the label. The options available are **Index**, **First**, and **Last**. This control works with the **Function Operator** and **Function Argument** fields to set the position of the label within the title block area.

Function Operator

Works with the **Function** and **Function Argumen**t fields to set the position of the label within the title block area. The value is controlled by the **Function** selection. This field is not editable.

Function Argument

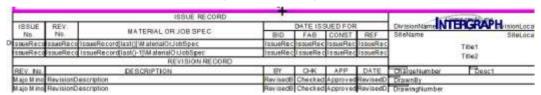
Sets a value to add or subtract from the **First** or **Last** settings in the **Function** field. This field works also works with the **Function Operator** field to set the position of the label within the title block area.

Alternative Text Value

Specifies alternative text to use if the selected property has no current value.

Display Label Names

Turns the label names on/off in the open border template:





Expands the ribbon to include additional formatting controls.



The controls on the expanded ribbon include:

Style

Sets the overall style used within the label.

Font

Sets the font for the label text.

Font Size

Sets the font size for the label text.

Textbox Width

Specifies the width of the text box.

Place a Drawing Property Label on a Template

If you want to place a custom attribute label on a template, see *Place a Custom Drawing Property Label on a Template* (on page 478).

1. Click Tools > Edit Border Template.

The Select Template dialog box displays.

2. Select a template, and click OK.

The template opens in SmartSketch Drawing Editor.

3. On the **Drawing Labels** toolbar, click **Place Drawing Property Label** ...

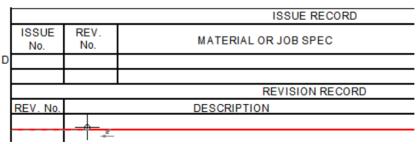
The Place Drawing Property Label ribbon displays.



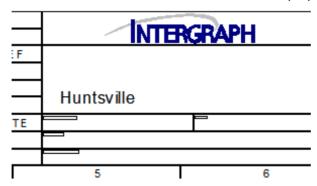
• TIP Click **Display Label Names** to show the labels names as they apply to the open border template:

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- 4. In the Label Set list (the first dropdown on the ribbon), select a label set (set of drawing properties). The list reflects the label sets within the drawing XML schema. The Label Set selection controls the contents of the Fields list and the enabling of other controls on the ribbon.
- 5. In the **Fields** list (the second dropdown on the ribbon), select a field to use as your title block label.
- 6. Specify a **Function**, setting the **Function Argument** if needed.
- 7. Provide alternative text for cases when the label property could be blank.
- 8. Click More to expand the ribbon and set formatting options. Select the Style, Font, Font Size, Text Color, Textbox Height, and Textbox Width.
- 9. Zoom into the area of the border where you want to place the drawing property label. Click the template to place the label.



10. Continue placing labels on the template as necessary. For example, the following border shows that the **General** label for the **Location** property has been placed as **Huntsville**.



- 11. Right-click to end the command.
- 12. Save the changes to the template before closing **SmartSketch Drawing Editor**.

See Also

Publishing Title Blocks (on page 519)

Place a Custom Drawing Property Label on a Template

The following steps show how to add a custom attribute drawing property label to a drawing template. To add non-custom property labels to the template, see *Place a Drawing Property Label on a Template* (on page 477).

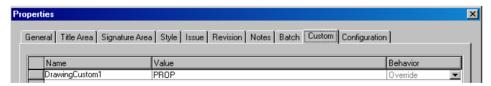
- 1. Create a Custom Attribute workbook using Excel. This is the bulkload file for the custom attribute and names the attribute. For example, you could create a file called *CustomAttributes.xls* that contains the definition for a custom attribute. For more information on the format for the workbook, see *Drawings-ExtendCustomAttributes Workbook* in the *Drawings and Reports Reference Data Guide*.
- Bulkload the Custom Attribute workbook. For more information on populating Excel workbooks and bulkloading, see the *Reference Data Guide* available from **Help > Printable Guides**.
- Create an .xsd file and add a line that defines the attribute in the dropdown list when you
 edit a template and use the Place Drawing Property Label command. For example, using
 DrawingCustom1 as the attribute name, the line would be <xs:element
 name="DrawingCustom1">.

NOTE You can only use letters, numbers, and underscores when defining the **element name**. The name must also begin with a letter.

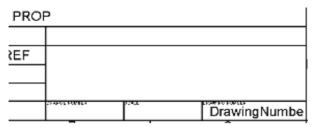
4. Add a line that points to the **AttributeName** property of the bulkloaded attribute. For example, using the above attribute name, the necessary line would be *<pk* name="DrawingCustom1"/>. Your .xsd file would look similar to the following:

NOTE The **pk name** in the Custom.xsd file must match the related **AttributeName** property of the bulkloaded attribute.

- 5. Make sure the .xsd file is in the \Symbols\Drawings\Catalog\Labels\Border\Schema folder.
- 6. In the Drawings and Reports task, click **Tools > Edit Border Template**.
- 7. On the **Select Template** dialog box, select a template, and click **OK**. The template opens in **SmartSketch Drawing Editor**.
- 8. In the **Label Set** list (the first dropdown on the ribbon), select **Custom**.
- 9. In the **Fields** list (the second dropdown on the ribbon), select the Custom attribute property to use as your title block label.
- 10. Zoom into the area of the border where you want to place the drawing property label. Click the template to place the label.
- 11. Continue placing labels on the template as necessary.
- 12. Right-click to end the command.
- 13. Save the changes to the template before closing **SmartSketch Drawing Editor**.
- 14. Update any drawing documents associated with the modified template. For more information, see *Updating Documents* (on page 82) in the *Drawings Help*.
- 15. Right-click the drawing and select **Properties**.
- 16. Go to the **Custom** tab. The new Custom attribute property is shown on the tab.



17. To verify the custom drawing property is added to the drawing title block, right-click the drawing and select **Open**. The custom drawing property includes the Custom Property attribute in the title block.



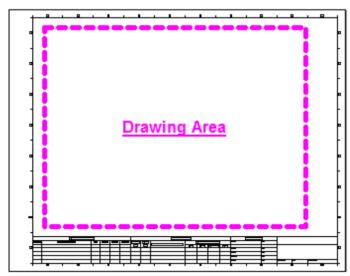
Place Drawing Area Command

Places a drawing area for drawing border templates. This command is available when you are editing a drawing border template for which a drawing area is not defined. If a drawing area already exists, an error messages displays. You can delete the existing drawing area and place a new one.

A drawing area is a property of the border template that defines the "useful" area of a drawing sheet. During the creation of a new composed drawing, the layout template containing the managed views fits inside the drawing area of the border template.

By defining a drawing area in each border template, a single layout template can be fit into any size border. The proportion of the drawing area devoted to a particular region in the layout does not change when the border template changes. For example, if you apply a region to the bottom half of a layout (and the layout is merged into a border template), the region always consumes the bottom half of the drawing, regardless of how it is stretched or resized by different border templates.

After you click **Place Drawing Area** , click and drag to place the drawing area.



Save the border template file and exit SmartSketch Drawing Editor. You can use the border file to create new 3D composed drawings. For more information, see the *Drawings and Reports User's Guide* or the *Common User's Guide*.

■ NOTES

- Editing a border template on the **Symbols** share does not affect an existing drawing because the border template associated to a drawing is stored in the database. To change the border template used by an existing drawing, you must replace the stored border template using the **Switch Border** command.
- If no drawing area exists in the border template used to create a composed drawing, one is computed based on the white space in the border template. This computed drawing area is not saved with the border template.
- The existence of a drawing area in the border template associated to a drawing has an impact on the behavior of the Switch Border command.

See Also

Switch Border (on page 485)

Change the Border for an Individual Drawings by Rule Sheet

The following procedure steps you through changing the border for an individual sheet to a template different from the border template defined in the Drawings by Rule component. For example, you may want the first sheet of drawing to use a different border template than all other sheets.

NOTE For more information on **SmartSketch Drawing Editor** commands used in this procedure, see the *SmartSketch Drawing Editor Help*.

Change the Border Template for an Existing Sheet

- 1. Right-click a drawings by rule document in the Detail View and select Edit.
 - The drawing opens in **SmartSketch Drawing Editor**.
- 2. Select a sheet tab at the bottom of the drawing view.
- 3. Click Edit Sheet Properties don the toolbar.
 - The Sheet Properties dialog box displays.
- 4. On the **Current Sheet** tab, select a new value for **Border Template**.
- 5. Click OK.

The new border template is applied to the sheet.

NOTE Layout Template is inactive, and cannot be changed for an existing sheet.

Apply a Border and Layout Template to a New Sheet

- 1. Right-click a drawings by rule document in the Detail View and select **Edit**.
 - The drawing opens in SmartSketch Drawing Editor.
- 2. Right-click a sheet tab at the bottom of the drawing view and select **Insert**.
 - A new sheet is created and the **Sheet Properties** dialog box displays.

- 3. On the Current Sheet tab, select a value for Border Template and Layout Template.
- 4. Click OK.

The border and layout templates are applied to the new sheet.

NOTE A sheet cannot be deleted unless all views on the sheet are removed first.

Edit Layout Template

Available when you click **Tools > Edit Layout Template** in the Drawings and Reports task.

Edit Border Family Command

Allows you to change the border family associated with the current layout template. This command is available when you edit a layout template.

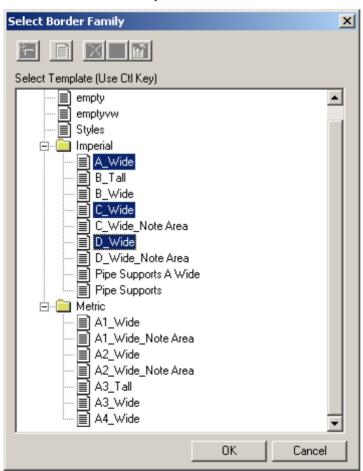
When you click **Edit Border Family** dialog box appears so that you can select border templates to associate to the current layout template.

See Also

Select Border Family Dialog Box (on page 483)

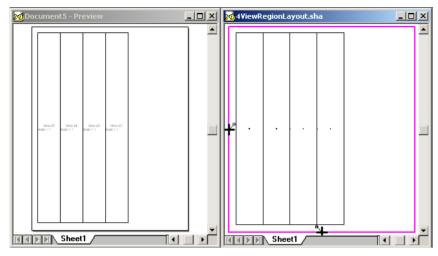
Select Border Family Dialog Box

Specifies the border templates to associate with the current layout template. This dialog box displays when you click **Edit Border Family**, which is only available in **SmartSketch Drawing Editor**. You can multi-select borders by holding down the CTRL key while selecting borders from the hierarchy.



Preview Layout Command

Shows a preview of the drawing document based on the current layout. A preview is shown for each of the borders chosen in the **Border Family** for the layout template.



This command is only available in **SmartSketch Drawing Editor** when you edit a layout template.

Shortcut Menus

Commands available when using right-click shortcut menus.

Update View Command

Updates the contents for the selected drawing, report, key plan, snapshot, or detail/section view. This command is available for composed drawing views only. Right-click a selected view and select **Update View** on the shortcut menu. The view contents update against the current 3D model. When working with the **Update View** command:

- If the software encounters a problem before or during the drawing update, it stops updating, displays either an error status or error message, and saves the errors to the log file. For more information, see Conditional Drawing Update (on page 91) in the Orthographic Drawings User's Guide.
- You cannot multi-select views. The command is not available on the shortcut menu if more than one view is selected.
- The inputs for the view (the associated volume or drawing view) must be defined and valid.
- If a report is defined as part of a drawing view definition, the Update View command is not available.
- If the drawing document is read-only, the **Update View** command is not available.

■ NOTES

- To remove associated inputs from a view, use the **Remove Associated Inputs** command. For more information, see *Remove Associated Inputs Command* (on page 396). To associate a view to a volume or another drawing view, use the **Associate Objects to View** command. For more information, see *Associate Objects to View Command* (on page 374).
- For composed drawings, views that are too small to display the volume are automatically sized larger on update to fit unless the view is set to **Fit to Scale** or is managed by a region. Likewise, views that are larger than the volume are automatically resized to fit unless the view is set to **Fit to Scale** or is managed by a region. The view size grows or shrinks from the center of the view and view proportions may change after the resize. The drawing must be saved to make the resize of the view permanent. For more information, see *Automatic Resize Behavior of Composed Views* (on page 358).

See Also

Place View Command (on page 347)
Associate Objects to Views (on page 390)
Place Snapshot View Command (on page 374)
Place Report View Command (on page 396)

Switch Border

Allows you to switch the border template associated with the composed drawing documents selected in the Detail View. This command is available on the shortcut menu when you select a composed drawing document. It displays the **Drawing Sheet Properties** dialog box so you can change the associated border template.

Editing a border template in the SharedContent folder does not affect an existing drawing because the border template associated to a drawing is stored in the database. To change the border template used by an existing drawing, you must replace the stored border template using the **Switch Border** command.

After switching the border template, the software computes a new position and size for any regions and managed views contained in the drawing. The shift and resize of the view is proportional to the size of the drawing areas in the border templates. If the border template does not contain a drawing area, the drawing boundary is computed automatically.

The software does not distinguish between different types of drawing views (report, key plan, and graphic views) when switching the border.

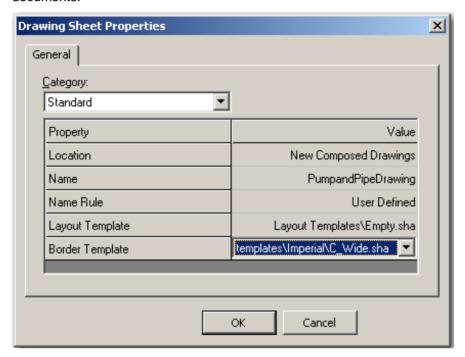
■ NOTES

- If you switch the border template of a drawing, views may resize or reposition with the new border template based on the following conditions:
 - Managed views are proportionately resized and repositioned according to the size of the new border template.
 - Unmanaged views are not resized and repositioned.
- Drawings must be updated after switching the border template in order to ensure all automated annotation is positioned correctly on the drawing.
- For views that are resized after the border template switch, drawing views using Fit to Scale show the same content. Views with a scale may have content clipped out if the view is made smaller after the switch.

For more information, see *Place Region Command* (on page 370).

Drawing Sheet Properties Dialog Box

Shows the drawing sheet properties associated with the selected drawing document(s) and allows you to change the border template file. When displayed for the **Switch Border** command on selected composed drawing documents, the only property available is **Border Template**. Select **More** in the **Value** field to display the **Select Template** dialog box and specify a new border template. Click **OK** to associate the new border template file to the selected drawing documents.



You need to update the drawings to regenerate them with the new border template file.

See Also

Switch Border (on page 485)

Add a Sheet to Drawing

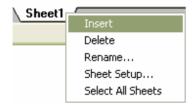
In this workflow, you change the border for an individual sheet to a template different from the border template defined in the component. For example, you may want the first sheet of drawing to use a different border template than all other sheets.

1. Right-click a drawings by rule document in the Detail View and select Edit.

The drawing opens in SmartSketch Drawing Editor.

2. Right-click a sheet tab at the bottom of the drawing view and select **Insert**.

A new sheet is created and the **Sheet Properties** dialog box displays.



- 3. On the **Current Sheet** tab, select values for **Border Template** and **Layout Template**. Click **Use Default** to apply the default values for the drawing, as shown on the **Default** tab.
- 4. Click OK.

The border and layout templates are applied to the new sheet.

Layers (SmartSketch Drawing Editor Tools Menu)

Layers are used to distinguish between graphics within a template or drawing. You can create layers in **SmartSketch Drawing Editor** with the **Tools > Layers** command.

Objects like drawing views, key plan views, report views, and drawing property labels are placed on the **DwgTemplate** layer when you save the drawing document. You should not place manual markups on the **DwgTemplate** layer.

When placing manual markups, such as graphics or company logos, place them in paper space within the drawing layers, not in *model space* (inside a drawing view). Markups placed within *model space* are not preserved. When a graphic rule does not specify a layer, intelligent graphics are placed on the **Default** layer.

Composed Drawings

When you create composed drawings, you can place manual markups on any layer of the drawing.

Volume (Spatial) and Orthographic Drawings by Query Drawings

For volume drawings, manual markups are preserved on any layer of the drawing, except the **DwgTemplate** layer. The **DwgTemplate** layer is reserved for system use. If you placed manual markups on the **DwgTemplate** layer, they will be lost when you update the drawing document.

Isogen Isometric Drawings

If you intend to create manual markups within an Isogen Isometric drawing, place your markups on the **Default** layer. The software preserves this layers when you update drawings. Other layers are not preserved.

If named layers do not exist in the template, the software creates them using the symbology specified in the style XML file. In ISOGEN Configuration, use the options in the **Drawing Area > Graphics > Layers** style group to create new layers within the style XML file. Map definitions to the layers under **Drawing Area > Graphics > Definitions**.

If the named layers do exist in the template, use **Tools** > **Display Manager** in **SmartSketch Drawing Editor** to change the symbology used within the template.

Modify an Existing Border File

You can create an isometric drawing border from scratch using **SmartSketch Drawing Editor**. You use the commands available within **SmartSketch Drawing Editor** to place graphics and create appropriate layers.

You should name new border files with the name of the needed isometric style, such as Iso Pipeline or Iso Piperun.

- 1. Navigate to [Reference Data Folder]\SharedContent\PmfglsoStyleData folder, and copy the appropriate existing border igr file to the [Reference Data Folder]\SharedContent\Drawings\Catalog\Templates folder. You can create a subfolder for the new file.
- 2. Rename the copied file with the .sha extension.
- 3. Open the copied .sha files with SmartSketch Drawing Editor.
- 4. Fit the view.
- 5. Select Tools > Layers.

The Layer ribbon displays.

- 6. Select **Tools** > **Display Manager**. On the **Layers** tab, scroll down to see the values currently set in the *.sha* file. These are the color, line type, and width values for the named layers.
- 7. Make changes as needed, save the file, and exit SmartSketch Drawing Editor. The next time you use this .sha file as your drawing border template, the graphics will show the changes you made.

See Also

Isometric Drawing Styles (on page 125) in the Isogen Isometric Drawings

Custom Commands

Custom commands used in SmartSketch Drawing Editor.

Convert Excel Spreadsheet Reports to Native Text Box Format Custom Command

The **SP3DConvertExcelEmbedded.dll** is a delivered custom command that allows you to convert an Excel spreadsheet report to the native text box format for use in 3D Drawings. The DLL to execute this command is located on the client machine in the [Product Folder]\Drawings\Client\Bin folder. To run this custom command, select **Tools > Custom Commands** in an open drawing in SmartSketch Drawing Editor, and browse to the DLL location on the computer. For additional information on any of the **SmartSketch Drawing Editor** commands mentioned in this procedure, see the *SmartSketch Drawing Editor Help*.

To convert an embedded report, you must set the properties correctly on the associated views. For a 3D composed drawing, set the **Report Output** property for a report view as needed. For more information, see *Report Properties Dialog Box (Place Report View Command)* (on page 398). For a 3D volume drawing, select the **Convert report output to text boxes (no Excel)** property for the drawing view associated to the report view. For more information, see *View Tab (Drawing View Properties Dialog Box)* (on page 467).

When you run this command, the **Convert** dialog appears so you can specify the appropriate options for the convert operation:

Convert embedded Excel spreadsheet

Indicates that the specified report will be converted to native text box format.

Convert and embed an Excel spreadsheet

Indicates that the specified report will be converted to native text box format and then embedded in the current drawing document.

Replace converted spreadsheet with a new converted version from Excel

Specifies that a converted spreadsheet report will be replaced with a new version of the same report.

File

Indicates the report file to convert. Click the ellipsis button to browse to the appropriate file.

See Also

Region Properties Dialog Box (on page 373)

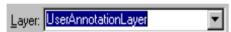
Edit a Composed Drawing

This procedure steps you through editing a composed drawing from either the Drawings and Reports task **Management Console** or from a 3D modeling task using the **Tools > Drawings Console** command. For additional information on any of the **SmartSketch Drawing Editor** commands mentioned in this procedure, see the *SmartSketch Drawing Editor Help.*

 Right-click an existing composed drawing in the Management Console or the Drawings Console. The document opens in SmartSketch Drawing Editor. You can use the editing tools in SmartSketch Drawing Editor to modify the appearance of your drawing. You can also use the Composed Drawing tools to place new regions, drawing views, report views, and snapshot views on the drawing area. You can also place labels manually on the drawing.

NOTE When you place objects like drawing views, key plan views, report views, and drawing property labels, the software automatically places them on the **DwgTemplate** layer when you save the drawing document. You should not place manual markups on the **DwgTemplate** layer.

If you use other native **SmartSketch Drawing Editor.** commands (such as **Place Line** or **Place Dimension**) to add manual markups to the template, put them on the **Default** or a layer with "User" as the prefix (for example, a layer named **UserAnnotationLayer** to preserve the changes when you update drawings. For more information on layers, see *Layers (SmartSketch Drawing Editor Tools Menu)* (on page 488).



- 2. To place a new region in the drawing area, use the **Place Region** command. Regions are used to manage drawing views. Click and drag to place the new region. The **Region Properties** dialog box displays after you place the region shape. Specify the properties for the region you are adding and click **OK** to complete the creation.
- 3. To place a new drawing view, use the **Place View** command. If you want your new region to manage this drawing view, place the new drawing views within or touching the region. You can also place "unmanaged" views outside existing regions. Click and drag to create the drawing view shape. The **Drawing View Properties** dialog box displays so you can define the drawing view. For more information on the **Drawing View Properties** dialog box, see *Drawing View Properties Dialog Box (Place View Command) Steel Order Drawings* (on page 354). For more information on how regions and views work together, see *Composed Drawings* in the *Orthographic Drawings User's Guide*.

4. To place a report view, click **Place Report View** . The drawing view you select can be managed by a region or unmanaged (outside all regions). Click and drag to place the report on the drawing area. The **Report Properties** dialog box displays so you can specify the properties for the report. Click **More** in the **Report Template** property box to select from available reports. You can set the output format of the report using the **Report Output Format** and **Report Justification** properties on this dialog box. For more information, see *Report Properties Dialog Box (Place Report View Command)* (on page 398).

■ NOTES

- The report view you create is empty. Use the **Associate Objects to View** command to associate a drawing view to the new report view. For more information, see *Associate Objects to View Command* (on page 374). The drawing view you select must already be associated with a volume in the model.
- You can associate multiple reports to the same view, but you cannot, in the current release, associate multiple views to the same report.
- 5. You can also place snapshot views on your composed drawing. You create the snapshot view content in a 3D task using the **Tools > Snapshot View** command. For more information on creating the snapshot view, see **Snapshot View Command** in the *Common User's Guide* available from **Help > Printable Guides**.
 - To place the snapshot view on the composed drawing, use the **Place Snapshot View** command. Click and drag to place the snapshot view shape. The **Drawing View Properties** dialog box displays so you can specify the properties for the snapshot view.
- 6. You can place manual labels on the composed drawing using the **Place Label** command. For more information on placing labels manually, see *Place a Manual Label* (on page 403).
- 7. If a view association is incorrect, click **Remove Associated Inputs** . This command allows you to select a view and remove all associations to volumes, folders, and other views.
- 8. If you decided to delete a drawing view, a message box displays when you press **Delete** for a selected view, providing a chance to cancel the operation. After a drawing view is deleted from the drawing, the operation cannot be reversed.
- 9. To update the contents of a view, select and right-click the view then select **Update View** on the shortcut menu. The view is updated with the associated 3D model objects or report, depending on the association you made with the **Associate Object to View** command.
- 10. Save your drawing changes before exiting **SmartSketch Drawing Editor**.
- 11. Update the modified drawing to incorporate the changes. You can open the drawing to check the new layout and view content.

Edit the Drawing Template

- 1. Right-click the component and select **Edit Template**. SmartSketch Drawing Editor and the **Sheet Properties** dialog box displays.
- On the Sheet tab, select values for Sheet Assignment Rule, Sheet Naming Rule, Border Template, and Layout Template. These values are used for the first sheet when a drawing document is created.
- 3. On the **Document** tab, select values for **Document Assignment Rule**, **Document Naming Rule**, **Border Template**, and **Layout Template**. These values are used for sheets added to a drawing document after it is created.
 - **NOTE** You can also click **Edit Sheet Properties** on the **Edit Template** ribbon toolbar to change values in the **Sheet Properties** dialog box.
- 4. Click File > Save.
- 5. Click File > Exit. The SmartSketch Drawing Editor window closes.

Modify View Ribbon

Modifies the view size. This ribbon displays after a view is selected in SmartSketch Drawing Editor.

Finish

Saves changes to the view.

Update View

Select to update the view (Drawing) gets updated along with the parent volume in the model.

Undo Crop

Select to remove an existing crop on the view. The view returns to its original boundaries. This option is not available for a view that cannot be cropped, such as a composed drawing view.

NOTE Removal of cropping with **Undo Crop** is not visible on a view until the view is updated. Select **Update View**, and click **Finish** to see the results immediately. The results will also be visible after using **Update** or **Batch Update** in the Drawings and Reports task at a later time.

Crop a Drawings by Rule 2D Drawing View and the 3D Model Volume

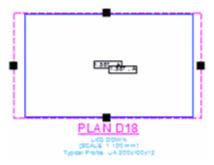
When an orthographic steel order view in a drawings-by-rule drawing is based on a volume in the model, you can crop the view, and also resize the model volume to match the cropped view.

1. Select the view in SmartSketch Drawing Editor.

Four handles display on the view frame and **Modify View** ribbon displays.



2. Drag the handles to resize the view to the needed size.



- 3. To immediately update the view, select **Update View**.
- 4. Click Finish.

The cropped view is saved and parent volume in the model is resized.

TIPS

- To cancel the changes to the view size, click outside of the view *without* clicking **Finish**.
- If Update View is not selected, the view is marked out-of-date in the Drawings View Explorer. You can right-click the view and select Update.

■ NOTES

- The 2D view and the 3D volume maintain the cropped size with subsequent updates.
- Shell expansions are not based on a volume and cannot be cropped. For more information, see Create a shell expansion drawing in the Orthographic Drawings User's Guide.
- Section and Detail views use their own commands for cropping. For more information, see *Place a Section View* (on page 436) and *Place a Detail View* (on page 440).

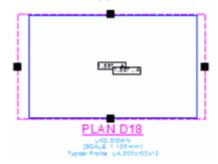
See Also

Drawings View Explorer (on page 324)

Remove Cropping on a Drawings by Rule View

1. Select a previously-cropped view in **SmartSketch Drawing Editor**.

Four handles appear on the view frame and Modify View ribbon displays.



2. Select Undo Crop and Update View.

3. Click Finish.

The view (and the parent volume in the model) changes to its original size.



Place an Unassigned View

The following steps describe the basic workflow for placing an existing view that is not assigned to a drawing.

- 1. Right-click a drawing document in the Detail View and select **Edit**. The drawing opens in **SmartSketch Drawing Editor**.
- 2. Drag an unassigned view from the **Unassigned Folder** in the Drawings View Explorer to the graphics view of the drawing.

The view displays in the graphics view and the view icon (or or or or or displays under the drawing sheet in the Drawings View Explorer.

■ NOTES

Dragging more than one view onto an existing drawing may require:

- Deleting the existing drawing region, and dragging the views onto the drawing without a region.
- Deleting the existing drawing region, and adding a new region that supports the number of views needed. For more information, see Edit Template (Drawings by Rule) in the Orthographic Drawings User's Guide and Layout Style Rules in Orthographic Drawings in the Drawings and Reports Reference Data Guide.

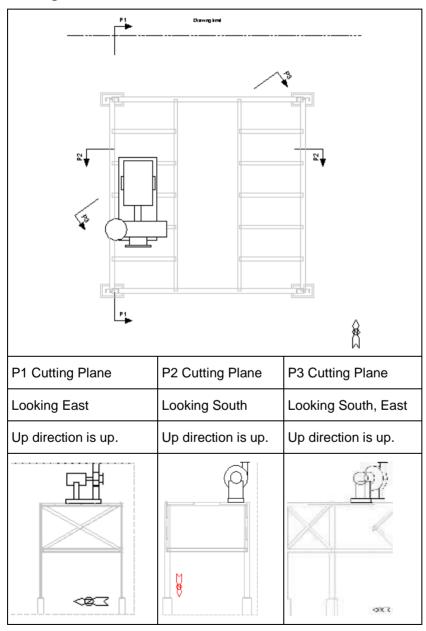
Section View Orientation Rules in Drawings by Rule

Plant Mode Section View Orientation Behavior

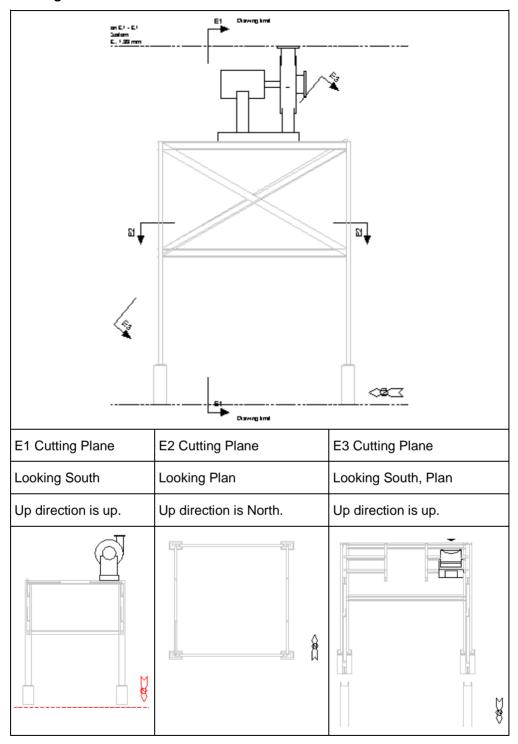
The software automatically determines the "up" direction for section views. In most cases, the up direction is "up," or +Z. In cases where using +Z is impossible (for example, if the section view direction is Looking Plan or Looking Up), the software orients the view to the North direction.

The following examples outline common section view orientations:

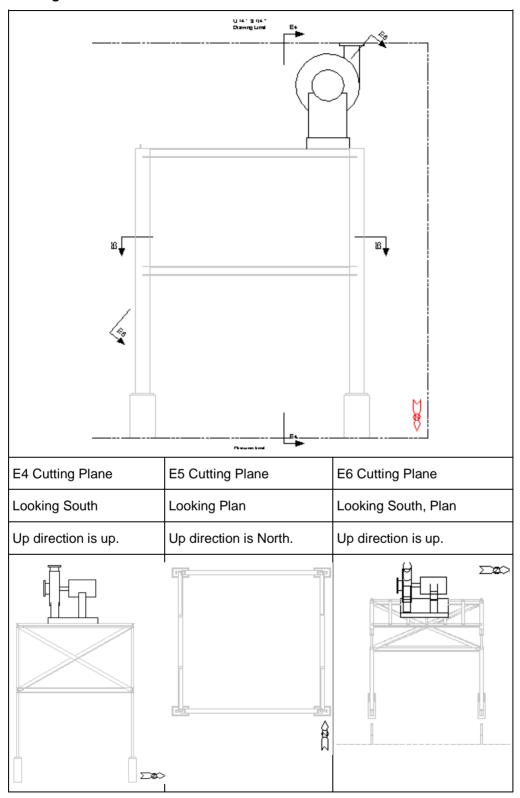
Looking Plan



Looking North



Looking East



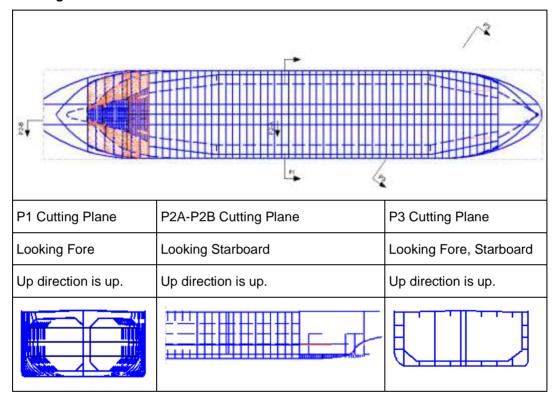
Marine Mode Section View Orientation Behavior

The software automatically determines the "up" direction for section views. In most cases, the up direction is "up," or +Z. In cases where using the z-axis is impossible (for example, if the section view direction is Looking Plan or Looking Up), the software orients the view to the Port direction.

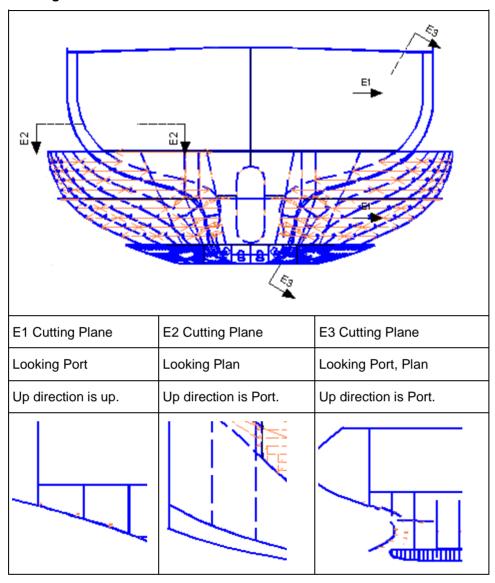
Additionally, angled sections of elevation views, and any section of a non-standard view direction (in which it is possible) uses Port orientation.

The following examples outline common section view orientations:

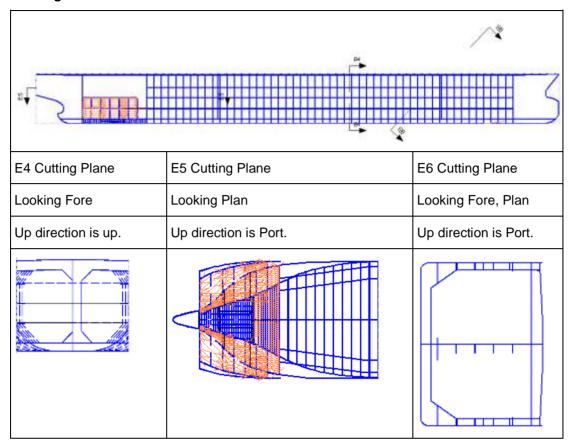
Looking Plan



Looking Aft



Looking Port



For more information, see *Orientation Rules in Drawings by Rule* in the *Drawings and Reports Reference Data Guide*.

Update and Full Update Commands

The **Update** and **Full Update** commands are available when you right-click a drawing view in the **Drawings View Explorer**.

Full Update performs a full update of all geometry in the view. It does not consider whether the model objects associated with the geometry has changed.

Update performs a smart update of the view or a set of selected views from the same sheet. If a set of criteria is met, then an incremental update is performed only for the geometry of added, modified, and deleted objects. If the criteria are not met, then a full update — the equivalent of **Full Update** — is performed.

All of the following criteria must be met for **Update** to trigger an incremental update:

- The view has had at least one full update previously performed.
- The only view properties that have changed are name and description. For more information, see View Tab (Drawing View Properties Dialog Box) (on page 349).
- The numbers of added, modified, or deleted objects are less than set quantities.
- The software allows incremental updates for the selected view type.

Any one of the following criteria cause **Update** to trigger a full update:

- The view is new and has never had a full update.
- The view is a section or detail view. For more information, see Place Detail Envelope Command (on page 423) and Place Cutting Plane/Section View Command (on page 426).
- The view has been cropped. For more information, see *Crop a Drawings by Rule 2D Drawing View and the 3D Model Volume* (on page 495).
- The view style, scale, orientation, or coordinate system have changed. For more information, see *View Tab (Drawing View Properties Dialog Box)* (on page 349).
- A primary plate object in the view has been spit or unsplit.
- The numbers of added, modified, or deleted objects are greater than a set percentage of the total number of objects.
- The software does not allow incremental updates for the selected view type.

■ NOTES

- An incremental update is usually faster than a full update, but still results in completely upto-date geometry for the view.
- The **View Log** command displays a log of the results from the last update performed on the view with the **Update** or **Full Update** commands.

See Also

Drawings View Explorer (on page 324)

Run Update or Full Update

You run **Update** or **Full Update** after structure displayed in a view has been modified in other tasks, such as Molded Forms or Structural Detailing.

1. Right-click a drawing document in the **Drawing Console** or the Detail View of the Drawings and Reports task, and select **Edit**.

The drawing opens in SmartSketch Drawing Editor.

2. If you know that objects in a view have changed, even if the view is still marked as up-to-date in the **Drawings View Explorer**, right-click the view in the **Drawings View Explorer** and select **Update** or **Full Update**.

A progress bar displays, detailing the update process.

NOTE You can update multiple views in the same sheet by selecting the sheet node or the individual sheets.

- 1. If you do not know if objects in a view have changed:
 - Right-click the view in the drawing area or the **Drawings View Explorer**, and select
 Refresh. The view is marked *out of date* in the **Drawings View Explorer** if objects have changed.
 - Right-click the view in the Drawings View Explorer, and select Update or Full Update.
- 2. Optionally, right-click the view in the **Drawings View Explorer**, and select **View Log** to check the results of the update.

Delete Views

Delete Views Placed by the Place View Command

- 1. Right-click a view placed by using Place View
- 2. Select Delete.

The **Delete Views** dialog box displays.

3. Select **Yes** to delete the view permanently. To cancel the operation, click **No**.

Delete Views Placed by Rules

- 1. Right-click a view in Drawing View Explorer.
- 2. Select Delete.

The **Delete Views** dialog box displays.

3. Select **Delete** to delete the view permanently.

■ NOTES

- The UnAssign option moves the view to the UnAssigned Folder. The Cancel option cancels the operation.
- If you delete a view that is a parent of other views, such as a detail or section view, the Convert or Delete dialog box displays. Select Convert to independent drawing view(s) to save the child view as an independent drawing view, or select Delete to delete the child view.

Delete Views from the UnAssigned Folder

- 1. Right-click a view from the **UnAssigned Folder**.
- 2. Select Delete.

The **Delete Views** dialog box displays.

3. Select **Yes** to delete the view permanently. To cancel the operation, click **No**.

SECTION 25

Save as SmartPlant Review File

You can view your 3D Model Data in SmartPlant Review (SPR) even if your model has not been registered using the SmartPlant Registration Wizard. When you create a 3D Model Data component, you set it up to save as a SmartPlant Review file. This enables the **Save As SmartPlant Review** command on the right-click menu for the 3D Model Data documents after they are created and up-to-date.

The **Save As SmartPlant Review** command is not available if you setup the 3D Model Data component to save to **Disk Only** or if you selected the **Generate CAD output (.sat file)** option on the **Setup** dialog box.

Before you can use the **Save As SmartPlant Review** command, you must install the SmartPlant Schema Component on the local client machine. For more information, see the *Intergraph Smart*TM *3D Installation Guide*, available from the **Help > Printable Guides** command.

SPR shows the objects from the VUE file using global coordinates. Before you save the 3D Model Data component documents as SmartPlant Review (SPR) files, right-click the component and select **Properties** to check the **Style** tab **Coordinate System** property setting. You want to make sure the **Plant Monument Coordinate Offset** is passed correctly to SPR when creating the VUE file. The offset value allows you to see the original coordinates relative to the new SPR coordinate system.

The 3D Model Data documents use surface styles and aspects as part of their property definition so that they can be used in SPR. You set the properties at the 3D Model Data document level. For more information, see *Set surface styles and aspects for 3D model data documents* (on page 47).

The **Save as SmartPlant Review** command saves a 3D Model Data document as a SmartPlant Review file. The file can then be viewed in SmartPlant Review. This command is available when you right-click an up-to-date 3D Model Data document if it was setup to save as a SmartPlant Review file. For more information, see Setup a 3D Model Data component. The **Save As SmartPlant Review** command is not available if you set up the 3D Model Data component to save to **Disk Only** or if you selected the **Generate CAD output (.sat file)** option on the **Setup** dialog box.

The command creates both a property data file in XML format and a graphics file in VUE format (.vue).

For more information, see Save as SmartPlant Review Dialog Box (on page 511).

Recommendations for Exporting to SmartPlant Review

- The number of objects generated by a 3D Model Data component and exported successfully to a SmartPlant Review file depends largely on the type of objects and your hardware resources. The more objects in your file, the more processing time the software requires. It is possible to have so many objects in the file that the software uses up all of the available memory. Larger models can take hours to process, so batch updating is recommended.
- Monitor the error logs regularly for resource issues, even if the specified filter worked initially.
 You can add more objects to the model meeting the filter criteria.
- SmartPlant Review (SPR) version 6.1.0.15 (or higher) allows you to open multiple VUE files simultaneously. Refer to your SmartPlant Review documentation for more information. When you open VUE and XML files in SPR for the first time, SPR builds a database containing the tag information for the files. This process may take a significant amount of time. After the multiple VUE files are opened in SPR, you can create SVF files for future loading of VUE files.
- SmartPlant Review (SPR) version 6.2.0.29 (or higher) supports turning Smart 3D aspects on and off. All aspects are turned on by default in SPR. The SPRSchema.txt file can be customized to add aspects.

■ NOTES

- You can use SmartPlant Review to review each resulting model of Save as SmartPlant Review action. However, you cannot review multiple models.
- Before you can use the **Save as SmartPlant Review** command, you must install SmartPlant Schema Component on the local client machine. For more information, see the *Intergraph Smart*TM 3D *Installation Guide*, available from the **Help > Printable Guides** command.
- The **Save as SmartPlant Review** command also looks for the EFSchema file (*P3DComponent.xml*). If your model has been registered using the SmartPlant Registration Wizard, the software retrieves the file from the integrated environment. If your model has not been registered using the SmartPlant Registration Wizard, the software looks for the file in the SharedContent share.
- You can publish .zvf files and launch SmartPlant Review to view the .zvf files. You can also view the .zvf file using View and Markup if you have SmartPlant Markup Plus installed. You do not have to use the Save as SmartPlant Review command to use the SmartPlant Review features.
- The **3D View Control** used for viewing the published graphics in SmartPlant Markup Plus does not currently support turning aspects on and off.

Save 3D model data for SmartPlant Review

Before you can use the **Save as SmartPlant Review** command, you must install the SmartPlant Schema Component on the local client machine. For more information, see the *Intergraph Smart*TM *3D Installation Guide*, available from the **Help > Printable Guides** command. The **Save as SmartPlant Review** command also looks for the EFSchema file (*P3DComponent.xml*). If your model has been registered using the SmartPlant Registration Wizard, the software retrieves the file from the integrated environment. If your model has not been registered using the SmartPlant Registration Wizard, the software looks for the file in the SharedContent share.

The **Save as SmartPlant Review** command is available on the right-click menu for the 3D Model Data documents when they are set up to save as a SmartPlant Review file. For more information, see Setup a 3D Model Data component.

Recommendations for Exporting to SmartPlant Review

- The number of objects generated by a 3D Model Data component and exported successfully to a SmartPlant Review file depends largely on the type of objects and your hardware resources. We recommend you limit each 3D Model Data component filter to approximately 30,000 objects if using Smart 3D version 6.1 Service Pack 1 or later (approximately 20,000 objects is using an earlier version).
- Monitor the error logs regularly for resource issues, even if the specified filter worked initially.
 You can add more objects to the model meeting the filter criteria.
- SmartPlant Review (SPR) version 6.1.0.15 (or higher) allows you to open multiple VUE files simultaneously. Refer to your SmartPlant Review documentation for more information. When you open VUE and XML files in SPR for the first time, SPR builds a database containing the tag information for the files. This process may take a significant amount of time. After the multiple VUE files are opened in SPR, you can create SVF files for future loading of VUE files.
- SmartPlant Review (SPR) version 6.2.0.29 (or higher) supports turning Smart 3D aspects on and off. All aspects are turned on by default in SPR. The SPRSchema.txt file can be customized to add aspects.

NOTE The **3D View Control** used for viewing the published graphics in SmartPlant Markup Plus does not currently support turning aspects on and off.

- 1. Before you save your 3D Model Data component documents as SmartPlant Review files, right-click the component and select **Properties** and go to the **Style** tab to make sure the **Coordinate System** property is set appropriately so that the **Plant Monument Coordinate Offset** is passed correctly to SPR when creating the VUE file. This is because SPR shows the objects from the VUE file using global coordinates. The offset value allows you to see the original coordinates relative to the new SPR coordinate system. For more information on 3D Model Data components, see 3D Model Data. For information on saving to SPR, see *Save as SmartPlant Review File* (on page 508).
- 2. You should also make sure that the **Surface Styles and Aspects** properties are set correctly on the 3D Model Data documents before saving them for viewing in SmartPlant Review.

Set surface styles and aspects for 3D model data documents (on page 47)

- 3. Right-click an up-to-date 3D Model Data document and select **Save As SmartPlant Review**.
- 4. On the **Save as SmartPlant Review** dialog box, specify a property data file. This file will be an XML format file.
- 5. Specify a graphics file. This file will be a VUE format file (.vue).
- 6. Click **OK** save the SmartPlant Review files to the specified names and locations.

Save as SmartPlant Review Dialog Box

Specifies the files needed to save documents for viewing in SmartPlant Review.

NOTE You can publish .zvf files and launch SmartPlant Review to view the .zvf files. You can also view the .zvf file using **View and Markup** if you have SmartPlant Markup Plus installed. You do not have to use the **Save as SmartPlant Review** command to use the SmartPlant Review features.

Data file

Specifies the XML file that contains all the objects and property data that the application publishes for viewing and that meet the filter query specified for the 3D Model Data component. For more information on the property data that is published, see *The Integration Reference Guide* located in **Help > Printable Guides**.

Graphics file

Specifies the VUE file (.vue) to which the document graphics are saved. This is the file you would select in SmartPlant Review with the **File > Open** command.

You can browse for either file using the ellipsis button next to the field.

See Also

Save as SmartPlant Review File (on page 508) Save 3D model data for SmartPlant Review (on page 509)

SECTION 26

Revising

The document revision process is separate from the publishing process, making it possible to revise a document locally and save it to the database without re-publishing it. The **Revise** command is available on the right-click menu for drawings, reports, and 3D Model Data documents. In an integrated environment, all revisions are handled by SmartPlant Foundation.

Revising and publishing are two separate actions. You specify the document revision using the **Revise** command, which creates a Revision for the document with Major and Minor set, depending on the revision schema selected. If you are working in an integrated environment, you can modify the other revision information on the document.

After setting the revision number, right-click the document and select **Properties**. Select the **Revision** tab and edit the **Revision** fields. You should update documents to include any new title block information.

You can now re-publish the document with the new revision information.

■ NOTES

- You can use the Revise command if your model has been registered using the SmartPlant Registration Wizard. For more information on registering, see the *Project Management User's Guide* under Help > Printable Guides.
- If the drawing document that you are looking at in the **Detail View** has a yellow icon (for example: , the drawing document is a version 6.1 legacy Snapshot drawing. You should use the **Tools > Convert Legacy Snapshots** command to convert this document to a Composed Drawing for use in the current version of the software. If you do not convert the legacy snapshot drawing, you cannot perform edit operations on the drawing, including update, revise, and publish.

See Also

Publishing Documents (on page 517) Revise (on page 513) Revise a document (on page 514)

Revise

Revising and publishing are two separate actions. You specify the document revision using the **Revise** command, which creates a Revision for the document with Major and Minor set, depending on the revision schema selected. If you are working in an integrated environment, you cannot modify the Major and Minor revision data, but you can modify the other revision information on the document.

After reserving the revision number, right-click the document and select **Properties**. Select the **Revision** tab and edit the **Revision** fields. You should update documents to include any new title block information.

You can now re-publish the document with the new revision information.

Revise Dialog Box (on page 515)

■ NOTE You can use the **Revise** command if you have registered your model using the SmartPlant Registration Wizard. For more information on registering, see the *Project Management User's Guide* under **Help > Printable Guides**.

Revise a document

You can revise drawings, reports, and 3D Model Data documents if you have registered your model using the SmartPlant Registration Wizard. For more information about using the SmartPlant Registration Wizard, see the *Project Management User's Guide* under **Help > Printable Guides**.

1. Right-click a document and select Revise. The Revise dialog box displays.

TIPS

- You can also multi-select documents in the **Detail View**, or you can select a folder in the **Management Console** hierarchy to select all of the documents within the folder if they all have the same revision level.
- If the Revise command is not available on the shortcut menu, check the properties on the document. Right-click the document and select Properties. Go to the WBS tab and make sure that you have a Document type and Discipline set for the document. For more information, see Set properties for publishing documents (on page 521).
- 2. For a new document or a document that does not yet have a defined revision scheme, select the revision scheme that you want to use from the **Revision Scheme** list.
 - PNOTE Only revision schemes that are applicable to the configuration (plant) or classification (document type) are available in the shortcut menu. The revision schemes related to a configuration or classification are not available for any other configurations or classifications. If none of the revision schemes are related to the configuration or classification, then all revision schemes are available unless they are related to any other configuration or classification. For more information on revision scheme configuration, see Configuring Different Revision Scheme Strategies in the How to Configure Document Management guide.
- 3. In the Revise in Tool section, select the next available major and minor revision numbers.
- 4. Click **OK**. The document is saved to the model database. The command creates a revision record by adding it to the document Revision properties. The command also reserves the revision number.

- 5. Right-click the document and select **Properties**.
- 6. Go to the **Revision** tab and edit the values in the new revision row.
- 7. Update the document to update any document property title block information. For more information, see *Updating Documents* (on page 82).
- 8. Re-publish the document. The stored document is not updated until you publish it.

Revise Dialog Box

Allows you to revise a document in the database of the authoring tool without publishing it.

INOTE Fields with a shaded background are read-only fields and cannot be edited.

Selected documents

Displays a list of the documents selected to be revised or for which you want to reserve a set of revision numbers. You populate this list by selecting documents before you use the **Revise** command.

Engineering Tool

Opens an authoring tool-specific dialog box that allows you to select documents to add to the **Selected documents** list. This option is not available in Smart 3D.

Revision Scheme

If you have selected a new document or a document for which no revision scheme has been selected, choose the revision scheme to be applied from the list of available options. Only revision schemes that are applicable to the configuration (plant) or classification (document type) are available in the shortcut menu. The revision schemes related to a configuration or classification are not available for any other configurations or classifications. If none of the revision schemes are related to the configuration or classification, then all revision schemes are available unless they are related to any other configuration or classification. For more information on revision scheme configuration, see *Configuring Different Revision Scheme Strategies* in the *How to Configure Document Management* guide.

Current Revision in Tool Major

For existing documents, this field displays the current major revision of the document, as defined in the authoring tool, in a read-only format. For new documents, this field is empty.

Current Revision in Tool Minor

For existing documents, this field displays the current minor revision of the document, as defined in the authoring tool, in a read-only format. If the revision scheme does not use minor revision, or if the selected document has not yet been revised, this field is empty.

Revise in Tool Major

From this list box, choose the next available major revision number for the document to revise it locally, without publishing the new information. If you do not want to revise the document at this time, in other words, if you want to reserve revisions numbers without revising the document, leave this field empty.

Revise in Tool Minor

From this list box, choose the next available minor revision number for the document to revise it locally, without publishing the new information. If you do not want to revise the document at this time, in other words, if you want to reserve revisions numbers without revising the document, leave this field empty. If minor revisions are not supported for the document, no options are available in this list.

★ IMPORTANT If you do not use the **Minor** field when revising a document for the first time, the minor revision option will never be available for that document for future revisions.

Next Major

Updates the document revision to the next major revision.

SECTION 27

Publishing Documents

When you work in an integrated environment with SmartPlant Enterprise, you must publish documents containing the drawing data and relationships before other authoring tools can share this information. You can publish your documents from the Drawings and Reports task **Management Console** or from a 3D modeling task by using the **Tools > Drawings Console** command.

Before you can publish documents in the software, you must install the SmartPlant Client and the SmartPlant Schema Component, then register the model using the SmartPlant Registration Wizard.

In a 3D task, you can use **SmartPlant > Retrieve** to create and update the Design Basis objects.

The software allows you to publish modified and new objects with the **Changes Only** option. Publish tasks processed through the **Changes Only** workflow are smaller compared to **All** publishes, and are queued to **Load** and **Consolidate** before the merge operation. The merge operation combines the delta data with the previous complete publish data. After the merge operation succeeds, the information is retrievable. The **Changes Only** publishes are not retrievable. To retrieve a **Changes Only** publish, you must perform an **All** publish, which only happens after the load and consolidate processes.

The **Publish** and **Update and Publish** commands are available for the following document types:

- 3D Model Data (SmartPlant Review file type)
 - **NOTE** The 3D Model Data component is capable of publishing many object types (for example: Piping, Equipment, Cable Trays) depending on the definition of the filter during component setup.
- Orthographic Drawings (viewable file with links to data)
- Isogen Isometric Drawings (viewable file with links to data)
 - NOTE The software supports publishing additional files (for example: PCF, POD) along with the SHA drawing. For more information, see *Enterprise Data* in the *Isometric Drawing Options Reference Guide*. You can access this document using the **Help > Printable Guides** command in the software.
- Reports (viewable file with links to data)

■ NOTES

- The viewable files created when you publish drawings and reports provide relationship links to the 3D Model Data. You must also publish the 3D Model Data to provide the navigation between the viewable files and the 3D Model Data.
- The Publish > Update and Publish command updates and then immediately publishes the selected documents in one step. This command is available only if the model is registered with SmartPlant Foundation.
- For a list of common tasks related to publishing, see *Publish Common Tasks* (on page 520).

- If you are publishing 3D Model Data documents, set the surface style rules and aspects before publishing the documents.
- Every time you generate drawings and reports from Smart 3D in an integrated environment, a SmartPlant Foundation token is used.

When you publish documents, the software:

- Publishes a visual representation of the document that you can view without Smart 3D. For drawings, this is an Intergraph proprietary file, called a RAD file (.sha). For reports, the viewable file is a Microsoft Excel workbook. You can review and mark up the visual representation of the document using SmartPlant Markup Plus or SmartSketch.
- Places the published XML file and any viewable files in the appropriate SmartPlant Foundation vault. This XML file can be retrieved when you are in other authoring tools.

Reasons to Publish

You publish documents and associated data for several reasons:

- Exchanging of data with other tools
- Sharing common data between tools
- Providing enterprise-wide accessibility to published documents
- Managing change, including workflow history, document revisions, and title block information

Revisions and Versions of Published Documents

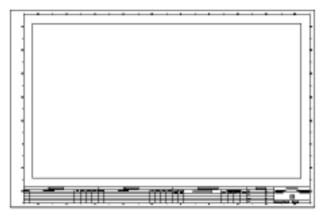
The first time that you publish a document, the software creates a new document master and the first revision. A revision (major) is an officially recognized change to a document. A version (minor) is an intermediate update that you have published. Revisions can be published for sharing or they can go through an approval process, depending on your needs. Each revisions of a document can have multiple versions.

You can also include revision information within the title block of a drawing by placing drawing property labels within the drawing template.

NOTE When you publish data from any authoring tool, you may not be able to view all of the properties that you published in the SmartPlant Client. You can customize view definitions to allow you to see additional properties. For more information on defining view definitions in the SmartPlant schema, see the *SmartPlant Schema Editor User's Guide*. For further assistance with viewing data, contact Intergraph Support Services. You can find support information on our web site, http://support.intergraph.com (http://support.intergraph.com).

Publishing Title Blocks

The title block is generally displayed at the bottom of a drawing template. It can include signatures, revision and issue information, and other properties associated with the drawing.



You add drawing properties, such as revision information or issue requests, to the title block using the **Place Drawing Property Label** command when editing a template.

When you update the revision information or receive an issue request on a drawing document, the associated properties must be updated. You update the drawing document to incorporate the property changes for any drawing property labels included in the title block.

After you update the drawing document, you can re-publish it and the appropriate title block information is recorded in SmartPlant Foundation. Publishing your drawing document helps you manage the changes, including workflow history, document revisions, and title block information.

The general workflow for including information in the drawing title block is:

- 1. Edit a template and include drawing property labels for revision or issue request *Place a Drawing Property Label on a Template* (on page 477).
- 2. Create drawing documents using the template.
- 3. Publish the drawings *Publish documents* (on page 526).
- 4. Revise the drawing document or receive an issue request from SmartPlant Foundation Revise a document (on page 514) or Issue request documents (on page 529).
- 5. Update the drawing documents to include the new revision, issue, or other drawing property information in the title block.

See Also

Delivered Drawing Types (on page 20) Publishing Documents (on page 517) Updating Documents (on page 82)

Publish Common Tasks

The following tasks are used to publish documents. If the documents are drawings or reports, the **Publish** command publishes a viewable file with links to the data. If publishing a 3D Model Data document, the software creates a SmartPlant Review file and publishes it.

■ NOTE The viewable files created when you publish drawings and reports provide relationship links to the 3D Model Data. You must also publish the 3D Model Data to provide the navigation between the viewable files and the 3D Model Data.

Setting Properties for Publishing

Before using the Publish command, you should set certain properties on your documents.

If you are publishing 3D Model Data documents, set the surface style rules and aspects as needed before publishing.

Creating and Reserving Revision Numbers

You should create your documents by right-clicking them and selecting **Create Document(s)**. After they are created, if you require revision numbers for the documents, use the **Revise** command to reserve the revision numbers.

Update the Documents

Update the documents, right-click the component and select the appropriate **Update** command. You can also use **Batch > Update** if you are configured to use a batch server.

Publish Data

If you have registered your model using the SmartPlant Registration Wizard, you can publish your documents for retrieval and use in other tools. You can use the **Publish > Update and Publish** command to update drawings and publish them in one step. You can also use the **SmartPlant > Find Documents to Publish** command to generate a list of documents that need to be published and to publish terminations for deleted drawings, isos, and so on.

Publish

Publishes the information in the selected documents. You can access the **Publish Documents** command by right- clicking a component or document.

The **Publish** and **Update and Publish** commands are available for the following types of documents:

- 3D Model Data (SmartPlant Review file type)
- Orthographic Drawings, including Volume and Composed drawings (viewable file with links to data)
- Isogen Isometric Drawings (viewable file with links to data)
- Reports (viewable Microsoft Excel workbook file with links to data)

■ NOTES

- The viewable files created when you publish drawings and reports provide relationship links to the 3D Model Data. You must also publish the 3D Model Data to provide the navigation between the viewable files and the 3D Model Data.
- The Publish > Update and Publish command updates and then immediately publishes the selected documents in one step. This command is available only if the model is registered with SmartPlant Foundation.

★ IMPORTANT When you publish a 3D model, you must now enable the **Scheduler** and **Loader** in SmartPlant Foundation to make the 3D model data document retrievable. The load, consolidate, and merge tasks must complete successfully before the 3D model document can be retrieved.

Define the **Discipline** and **Document Type** properties to enable publishing for the documents. For more information see *Set properties for publishing documents* (on page 521).

NOTE For 3D model data, the **Discipline** and **Document Type** properties are already populated.

You may also want to specify documents to be revised, not published, or reserve revision numbers. For more information, see *Revising* (on page 513).

To generate a list of documents that need to be published, you can use **SmartPlant > Find Documents to Publish**. For more information, see *Find Documents to Publish* (on page 536).

You can publish isometric drawings to SmartPlant Foundation in additional file formats other than the **SHA** drawing file format. These additional files along with the **SHA** drawing are generated by the update process. When publishing isometric drawings you can also publish other available data files and reports generated by the update process. For more information, see Published Files (S3D Supplementary).

Publish Dialog Box (on page 530)

Set properties for publishing documents

- In the Console, right-click an item in the hierarchy, then select Properties on the shortcut menu.
- 2. Go to the WBS Tab.
- 3. Set the **Document Type** property as needed, such as Civil Plan. This property specifies the document subtype when published.
- 4. Set the **Document Style** property as needed, such as Ortho for an orthographic drawing.
- 5. Set the **Discipline** property. If your model has been registered using the SmartPlant Registration Wizard, this property adds the **Publish** command to the right-click menu for the selected document or documents. For a 3D Model Data document, the property is set to **SmartPlant Review Document**. For a drawing or report document, set the discipline to match the type of document.
- 6. Set the Allow Publish property to Yes.

■ NOTES

- If you do not want an item to acquire a property from its parent, select the **Override** column on the **Properties** dialog box, then type a new value. This value propagates to other items deeper in the hierarchy.
- The software treats blank or cleared property values as overrides.
- Before you can publish documents, you must:
 - Install the SmartPlant Client and the SmartPlant Schema Component.
 - Register your model using the SmartPlant Registration Wizard.
 - Revise and update each document.

For more information about configuration, see the *Intergraph Smart*[™] 3D *Installation Guide*, available from **Help > Printable Guides**.

Support for Handling Large Publishes

The number of objects published from Smart 3D can become so large that the normal update and publish processing uses up the resources such as memory and address space on the client and server computers. To address these resource issues, Smart 3D now supports the concept of using a cache to keep track of objects that have changed and to only publish these objects. Thus, the software avoids processing objects that have not changed. The mechanisms for doing this are called parallel update, delta publish, and auto-scoping. These processes are not visible to the user.

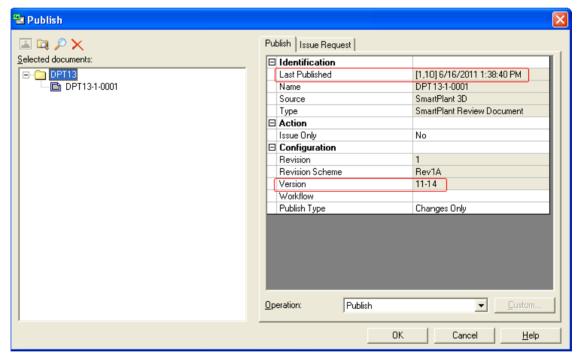
 Parallel update - Updates all the objects. The update process determines the number of processors on the system and uses this information to partition the objects so that multiple updates can happen at the same time. Because the updates are happening in parallel, the system can update a larger number of objects faster. Delta publish - Leverages the cached information provided by parallel update to determine which objects must be published. Because only changes are published, you can produce larger files in the data warehouse.

Because the complete set of objects will not be published each time, it is up to Smart 3D to keep track of deleted and moved objects. These moved or deleted objects will be sent with the published objects, eliminating the need to probe the adapter for the existence of missing objects.

★ IMPORTANT When you publish a 3D model, you must now enable the **Scheduler** and **Loader** in SmartPlant Foundation to make the 3D model data document retrievable. The load, consolidate, and merge tasks must complete successfully before the 3D model document can be retrieved.

Both parallel update and delta publish can publish larger documents by automatically splitting the data into smaller sections in Smart 3D. If your publish data is split into smaller sections, the document version number increments for each section.

For example, if you publish a document that is split into four smaller sections, the **Version** field contains all resulting version numbers. In the following dialog box, the published documents are published to SmartPlant Foundation as 4 different versions (11-14). The last version contains all of the updated information from the publish.



Published Documents - Load, Consolidated, and MergeDelta Tasks (S3D)

A tool published document results in a document version object in SmartPlant Foundation. A document version has four files attached to it.

- 1. **Data XML file** Contains published objects, relationships and correlations (SameAs relationships).
- 2. Meta data XML file Contains meta data about the published document version.
- Instructions XML file Contains instructions for deleted objects and relationships. Also
 contains resurrect instructions for objects that were deleted previously and resurrected as a
 result of restoring a tool database.
- 4. **View file** This is a file in the tool format that represents the (graphical) view of the published document.

When a document is submitted to a workflow that has a load process step, the following tasks are created.

NOTE If the document is not published to such a workflow, and when the **Load document** command in SmartPlant Foundation Desktop Client is run on the published document, the same tasks are created.

- Load Task This task processes the objects and relationships in the data XML by loading them into a publish domain. This task also processes the instructions in an Instructions XML file. For example, the delete instruction would result in termination of the object referenced in the delete instruction.
- Consolidate Task This task processes the correlations (SameAs relationships) in the data XML file. A SameAs is a correlation relationship published by tools indicating that a published object is identical to an object published by another tool. The correlation of the object in the current publish domain (say, local object) to the external object (object published by another tool in another publish domain) is done by creating a shared object in the Data Warehouse domain. The local and external objects are linked through SPFComprisedOf relationships; for example, one SPFComprisedOf relationship between the local object and the shared object, and a second SPFComprisedOf relationship between the external object and the shared object.

NOTE An object that is not correlated also has a shared object created in the Data Warehouse domain in preparation for future correlation.

The consolidate task also creates a hidden file, known as graphical mapping file, which has mappings between objects in the database and graphical elements in the view file. This file is used by SmartPlant Foundation Desktop Client for navigation between published objects in the list view and their graphical representations in the view file.

- MergeDelta Task This is a task created for delta publishes of Smart 3D. A delta publish contains new, modified and deleted objects in a document since the last publish of that document. Only Smart 3D has the capability to do a delta publish. Other tools publish full data where every object is published whether it is modified or not since last publish.
 - Because a delta published data XML file and view file only contains the delta, it is required that it becomes a full data XML and full view file for complete view of the data and graphics of that document. The MergeDelta task converts the delta data XML file into a full XML file by merging data from the previous full XML file into the delta XML file.
 - ★ IMPORTANT When you publish a 3D model, you must enable the Scheduler and Loader to make the 3D model data document retrievable. The load, consolidate, and merge tasks must complete successfully before the 3D model document can be retrieved. For more information, see Schedulers and Queue Management.
 - **NOTE** If you are using **SmartPlant Basic Integrator**, only the merge task must complete successfully before the 3D model document can be retrieved. For more information, see Schedulers and Queue Management.

The merge involves the following:

- a. Replace modified objects and relationships in the full publish XML, view file, and the hidden graphic map file.
- b. Delete objects and relationships that are identified in the delete, and move instructions in the delta instructions XML file, view file, and the hidden graphic map file.
- c. Add new objects and relationships to the full publish XML, view file, hidden graphic map file.
- d. Replaces the delta XML file in the vault with the updated full publish XML file, view file, and the hidden graphic map file.

Set SmartPlant Foundation to Tolerate Failed Loads

For working in any integrated environment, a new site configuration flag in SmartPlant Foundation Server Manager allows the SmartPlant Foundation Loader to skip a published document that failed during load and continue processing other documents in the queue.

In SmartPlant Foundation Server Manager, set the **SkipFailedDocumentsInLoad** flag to **True** to allow failed documents to be skipped and other documents to continue processing. Typical behavior is that after a document is successfully loaded, any previous failed document is removed from the queue.

The behavior during a Smart 3D publish is different. If any Smart 3D publish, either a full publish or delta publish, fails to load, the failed versions remain in the load queue until a full publish successfully loads. If a delta publish loads successfully following these failures, it does not remove any previous failed version from the queue because the data in delta publishes is different from version to version, so a failed previous version needs to stay in the queue until you resolve the failure or a full publish is loaded. A second reason is this gives an indication to you that a full publish is needed. When a full publish is loaded, all failed delta publishes are removed from the queue.

Publish documents

Before you can publish documents, you must ensure that your computer is configured properly. The configuration includes installing the SmartPlant Client and the SmartPlant Schema Component and registering the model through the SmartPlant Registration Wizard.

You must use the **SmartPlant > Retrieve** command in one of the 3D tasks to import published data

★ IMPORTANT When you publish a 3D model, you must now enable the **Scheduler** and **Loader** in SmartPlant Foundation to make the 3D model data document retrievable. The load, consolidate, and merge tasks must complete successfully before the 3D model document can be retrieved.

NOTE The **SmartPlant** menu is not available in all tasks.

1. Right-click a component and select **Publish**. The **Publish** dialog box displays.

■ NOTES

- If the Publish command is not available on the shortcut menu for the component or document, check the document properties and make sure that the documents are up-todate and have been revised first.
- You can use the Publish > Update and Publish command to update drawings and publish them in one step. This command is available only if the model is registered with SmartPlant Foundation.
- 2. Edit information as necessary for the selected documents.

When multiple documents are selected, only property values shared by all of the selected documents display in the table. Changing a value in the table changes that value for all of the selected documents.

- 3. Select the **Publish Type**.
 - Smart 3D supports Publish Type for delta publish.

■ NOTES

- The default setting for **Publish Type** is **Changes Only**. If the 3D model has not been published before, the software automatically selects **All** and performs a complete publish to ensure all filtered data is published and loaded into SmartPlant Foundation.
- For subsequent publishes, select All to publish all filtered data in the 3D model, even if it is unchanged. Select Changes Only to publish only the changes in the 3D model since the last successful publish.
- If the 3D model does not publish successfully during a delta publish, select All so that data from the previous publish is published and loaded into SmartPlant Foundation.
- Other SmartPlant applications do not support Publish Type. For these SmartPlant applications, the default setting for Publish Type is All, and it cannot be changed.

- 4. Select one of the following operations:
 - Publish to publish the selected documents immediately.
 - Background publish to publish as a separate process so that you can continue working in the application.
 - Scheduled publish to publish in the batch mode by the authoring tool. The documents
 are not published immediately. Instead, the selected documents are scheduled for
 publish at a later time and may be scheduled as a recurring operation.
- 5. Click **OK** to publish the selected documents.

■ NOTES

- You can verify the publishing process by starting the SmartPlant Client on your computer and searching for the published document.
- When publish is complete, the following message displays: Documents have been published successfully. If the View Log button is enabled, messages are available concerning the operation. These messages include errors, warning, and informational messages. Click View Log to review these messages.

Publish Workflows

When a SmartPlant application publishes, the user can publish documents using a workflow. The following list describes the delivered publish workflows and use cases for the workflows.

Workflow	Workflow Description	Workflow Use Cases		
Auto-Correlate	Compares the objects in a published document to the objects that are already in SmartPlant Foundation. If the object correlates (by name) to an object with the same shared object definition, SmartPlant Foundation automatically correlates the objects. If the object can correlate to more than one object through multiple shared object definitions, a task is placed in the To Do list to allow the user to select the correlation.	 When a publishing tool does not have correlation functionality. When a publishing tool does not retrieve. When using SmartPlant Foundation to perform consistency checking and the authoring applications do not retrieve. When Shared Objects To Correlate option is used to configure which types of objects are auto-correlated. 		
AutoLoadPublish	Generates a document load task and consolidate task for the publishing document. These tasks are performed by the SmartPlant Foundation Scheduler.	Use AutoLoadPublish when you intend to load the published document data into SmartPlant Foundation.		

Workflow	Workflow Description	Workflow Use Cases	
AutoLoadPublish Merge	Publish large 3D models. Creates the published 3D document, loads the 3D data in SmartPlant Foundation, then merges these object sets into a new composite document that represents the entire 3D model.	Use AutoLoadPublishMerge when you intend to view or navigate multiple 3D models as one composite 3D model document in one 3D view.	
Correlate	Places published objects with possible correlations in the To Do list. From the To Do list, the user manually correlates selected items. SmartPlant Foundation creates a shared object.	Correlate works similar to Auto- Correlate except you can view the possible correlations before the actual correlation takes place.	
Publish	Takes the document through an approval step. Upon approval, it creates the document load and consolidate tasks.	Use Publish when you want the published document approved before it is loaded into SmartPlant Foundation. The approval step is added to the To Do list.	
PublishWithDoc Load	Loads document into a document configuration and submits document to approval step. Upon approval, it creates document the load and consolidate tasks.	Use PublishWithDocLoad when you want to view the document data in SmartPlant Foundation before approval. The approval step is added to the To Do list.	
PublishForLoad Only	Generates a document load task and consolidate task for the publishing document and makes the documents irretrievable by any tool. These tasks are performed by the SmartPlant Foundation Scheduler.	Use PublishForLoadOnly when you want to publish documents to view, navigate and also for consistency checking in SmartPlant Foundation without the possibility of the document being retrieved by any tool.	

Publish a large 3D model to SmartPlant Foundation

Publishing large 3D models to SmartPlant Foundation has reached memory and file transfer limitations. To make publishing 3D models more efficient, you can publish models in distinct object sets that accommodate your memory resources. Distinct object sets mean that an object in the 3D model must appear in only one published model. You publish these object sets using a workflow called **AutoLoadPublishWithMerge**. This workflow has **MergePublishedFile** process step configured. This workflow creates the published 3D document, loads the 3D data in SmartPlant Foundation, then merges these object sets into a new composite document that represents the entire 3D model. The name of the composite document is same as the name of the workflow to which the Published 3D model is submitted. If you have existing published 3D models in SmartPlant Foundation and want to merge these into one representative model document, you can create a 3D composite document manually, and then relate the published models to the document.

- 1. Create filters for segregating and selecting data that will be in the published model documents. For example, you can create filters for equipment, piping, structural, and HVAC.
- 2. In the **Drawings and Reports** task, create your separate model documents.
- 3. Publish each model document separately.
 - **NOTE** The software publishes a new version of the 3D document, for example, a model document called Equipment-1-001, to SmartPlant Foundation. This document is related to **AutoLoadPublishWithMerge**, a zvf file, and a mapping file.
- On the Publish tab on the Publish dialog box, select the AutoLoadPublishWithMerge from the Workflow list.

Issue request documents

Before you can publish documents, you must ensure that your computer is configured properly. The configuration includes installing the SmartPlant Client and the SmartPlant Schema Component and registering the model with the SmartPlant Registration Wizard. For more information, see the *Intergraph Smart*TM 3D *Installation Guide* available from **Help > Printable Guides**. For more information on the steps involving the SmartPlant Foundation Desktop Client, see the *SmartPlant Foundation Desktop Client User's Guide*.

NOTE This functionality is only available in project mode. It is not supported in integration mode.

You must use the **SmartPlant > Retrieve** command in the Common task to import published data.

- 1. Right-click a component and select **Publish**. The **Publish** dialog box displays.
 - **NOTE** If the **Publish** command is not available on the shortcut menu for the component or document, check the document properties and make sure that the documents are up-to-date. For more information, see *Set properties for publishing documents* (on page 521).
- 2. Click the Issue Request tab.
- 3. In the **Issue to** field, select the contract that you want to assign the document or documents.
- 4. Under **Selected documents**, select the documents that you want to associate with the specified contract.

- 5. Click Add to add the documents to the Issue Request list. To remove documents from the list, select them and click Remove. Click Engineering Tools to add documents from engineering tools, such as P&IDs or PFDs. Click File System to add documents from another file system, such as Microsoft Word documents or Microsoft Excel workbooks.
- 6. Click **OK** to issue the contract request for the selected documents.
- 7. Start SmartPlant Foundation Desktop Client on your computer and search for the published document to verify the publishing process.
- 8. Right-click the document in the Desktop Client tree view and select Refresh.
- 9. Return to the Drawings and Reports task and update the document to incorporate the new Issue information. For more information, see *Updating Documents* (on page 82).
- 10. Review the Issue properties. Right-click the document and select **Properties**. Select the **Issue** tab to see the Issue information. You can also open the document to see the Issue information in the title block if you added it.
- 11. Publish the document with the updated Issue information. For more information, see *Publishing Documents* (on page 517).

■ NOTES

- Only updated documents can be published.
- You can verify the publishing process by starting the SmartPlant Client on your computer and searching for the published document.

See Also

Find Documents to Publish (on page 536) Publish (on page 520) Publish documents (on page 526) Publishing Title Blocks (on page 519)

Publish Dialog Box

Provides a list of documents selected to publish.

Publish Tab (Publish Dialog Box) (on page 531) Issue Request Tab (Publish Dialog Box) (on page 534)

See Also

Publish documents (on page 526)
Find Documents to Publish (on page 536)
Publish (on page 520)
Document Properties Dialog Box (on page 535)

Publish Tab (Publish Dialog Box)

Displays the properties of the selected document or documents. If only one document is selected in the tree view, the properties displayed on this tab are the properties of that one document. If multiple documents are selected, only the properties with the same value for all documents display. Any properties with varying values across the documents display with blank values in these fields.

You can change some of the values assigned to one or more documents by changing the value displayed in the table. The value you type here overrides any existing values for all selected documents.

Selected documents

Displays a list of the documents selected for publishing. You must populate this list by selecting documents in the **Management Console** or **Detail View** before you use the **Publish** command. For each document, this list displays the name, the type of document, the workflow from which the document was last published, the revision and version numbers, the revision scheme, and the date when the document was last published.

Engineering Tool

Opens a dialog box to select documents to add to the **Selected documents** list. This functionality is not available in the current release.

File System

Opens a standard Microsoft dialog box that allows you to select documents to add to the **Selected documents** list. When you select a file with this **Select File** dialog box, the **Document Properties** dialog box displays, allowing you to specify information about the file, such as whether it is a new file; the category, type, and subtype of the document; and the name, description, and title of the document.

Find

Opens the **Find Documents to Publish** dialog box, which allows you to search for documents to add to the **Selected documents** list. For more information, see *Find Documents to Publish Dialog Box* (on page 538).

Last Published

Indicates the date on which the document or documents were last published.

Name

Displays the name of the document.

Source

Indicates the authoring tool in which the document was created.

Type

Displays the type of document or documents selected.

Comment

Allows you to type information about the selected documents that are publishable.

Issue Only

Allows you to issue request documents without, necessarily, republishing them. Use this option when no changes were made to a drawing and you only want to add it to a contract.

TIPS

- Even with this option set, you can still publish the documents. If any of the documents have never been published, they must be published, regardless of this setting.
- You will receive an error message if you select multiple documents and activate this option when one or more of the selected documents cannot be changed. For example, the error message displays if the selected set of documents includes both a new document (for which this field can be set only to No) and current or locked documents (for which this field can be set only to Yes). The error message prompts you to select a smaller set of documents.

Owning Group

Select an owning group from the drop down list to which the document belongs.

■ NOTES

- By default, the owning group selected for the previous version, if any, is shown.
- All the owning groups configured in SmartPlant Foundation are listed.

Revision

Displays the current revision number of the selected document or documents.

TIP You will receive an error message if you attempt to change the value in this field when you have selected one or more documents that have conflicting revision schemes or different possible revisions. The error message prompts you to select a smaller set of documents.

Revision Scheme

Displays the revision scheme applied to the selected document or documents.

- NOTE Only revision schemes that are applicable to the configuration (plant) or classification (document type) are available in the shortcut menu. The revision schemes related to a configuration or classification are not available for any other configurations or classifications. If none of the revision schemes are related to the configuration or classification, then all revision schemes are available unless they are related to any other configuration or classification. For more information on revision scheme configuration, see Configuring Different Revision Scheme Strategies in the How to Configure Document Management guide.
- TIP You will receive an error message indicating that this field cannot be edited if one or more of the documents that you have selected are not new or will have a revision scheme supplied by the authoring tool. The error message prompts you to select a smaller set of documents.

Version

Indicates the current version of the document or documents.

Workflow

Indicates the workflow to which the selected document or documents are assigned.

Publish Type

Allows you to publish all data in a 3D model or only the changes to the model since the last successful publish. If the 3D model has not been published before, the software automatically selects **All** and performs a complete publish to ensure all filtered data is published and loaded into SmartPlant Foundation.

Publishing documents	Publish Type	
	All	Changes Only
First publish after creating the document	All objects	All objects
Subsequent publish with no changes to the model	All objects	No objects
Subsequent publish with changes to the model	All objects	Changed objects
Subsequent publish with no changes to the model (after changing the Publish Type)	All objects	No objects
Subsequent publish with changes to the model (after changing the Publish Type)	All objects	Changed objects

TIP You will receive an error message indicating that this field cannot be edited if one or more of the documents that you have selected have conflicting sets of possible workflows. The error message prompts you to select a smaller set of documents.

Check and publish released claims for previously deleted items

Specifies that you want to resolve issues where deleted items were restored from an earlier version and the claim on them was released. This check takes additional time and should only be used when deleted items have been restored. This option is not supported in this release.

TIP This check box should also be activated when publishing after a backup is restored or when releasing the claim on an object forces another tool to release the claim on a related object that was previously deleted. In this specific case, the tool fetches the object from As-Built again and releases the claim.

Operation

Specifies the operation to perform on the selected documents.

- Publish Selected documents are published immediately.
- Background publish Selected documents are published immediately as a separate process, allowing you to perform other tasks at the same time.
- Scheduled publish Selected documents are published in the batch mode by the authoring tool. This option is available only for tools that support batch mode and are processed by the authoring tool, not the SmartPlant Client. The documents are not published immediately. Instead, the selected documents are scheduled for publish at a later time and may be scheduled as a recurring operation.

NOTE If the software cannot make a SmartPlant Foundation server connection when you use **Scheduled Publish**, you are prompted to provide a valid SmartPlant Foundation login and password.

Custom

Opens the **Custom** dialog box. This functionality is available only if defined by your project implementation team.

Check for deleted objects no longer on documents

Select **Check for deleted objects no longer on documents** option if you want to process the move instructions while publishing.

See Also

Publish documents (on page 526)
Find Documents to Publish (on page 536)
Publish (on page 520)
Document Properties Dialog Box (on page 535)

Issue Request Tab (Publish Dialog Box)

Displays the documents associated with a specific issue request and allows you to add documents to or remove documents from a request.

Selected documents

Displays a list of the documents selected for publishing. You must populate this list by selecting documents in the **Management Console** or **Detail View** before you use the **Publish** command. For each document, this list displays the name, the type of document, the workflow from which the document was last published, the revision and version numbers, the revision scheme, and the date when the document was last published.

Engineering Tool

Opens a dialog box to select documents to add to the **Selected documents** list. This option is not supported in this release.

File System

Opens a standard Microsoft dialog box that allows you to select documents to add to the **Selected documents** list. When you select a file with this **Select File** dialog box, the **Document Properties** dialog box displays, allowing you to specify information about the file, such as whether it is a new file; the category, type, and subtype of the document; and the name, description, and title of the document.

Find

Opens the **Find Documents to Publish** dialog box, which allows you to search for documents to add to the **Selected documents** list.

Issue to

Contains a list of all objects (contracts) that can support issue requests. When you select an item from this list, the names of any documents associated with that object display in the table.

Add

Creates a new item in the table for any documents highlighted in the **Selected documents** tree view.

Remove

Deletes a selected document from the table.

Document Name

Displays the names of all documents associated with the object in the Issue to field.

See Also

Publish documents (on page 526)
Find Documents to Publish (on page 536)
Publish (on page 520)
Document Properties Dialog Box (on page 535)

Document Properties Dialog Box

Provides details about a new or existing document selected for publishing. You can access this dialog box using the following procedure:

- 1. Click **File System** on the *Publish Dialog Box* (on page 530). A Microsoft standard **Select File** dialog box displays.
- 2. Select a file to display the **Document Properties** dialog box and specify information about the file.
- TIP An asterisk (*) next to a field indicates that the field must be completed before the software enables the **OK** button.

Selected file

Displays the name of the file that you selected on the **Select File** dialog box.

New document

Indicates that this document has not previously been published.

Published previously

Indicates that the file has already been published at least once before.

Document category

Select the category for the document.

Document type

Select the type of document. The options that display in this list are determined by the selection that you make in the **Document category** field.

Document subtype

If applicable, select the subtype for the document. The options that display in this list are determined by the selection that you make in the **Document type** field.

Name

Type the name of the file as it will be known in the integrated environment.

Descriptions

Type a brief description of the file. This description displays later to help you recognize the file.

Title

Type the official title of the document.

See Also

Find Documents to Publish Dialog Box (on page 538) Publish Dialog Box (on page 530) Issue Request Tab (Publish Dialog Box) (on page 534) Publish Tab (Publish Dialog Box) (on page 531)

Find Documents to Publish

Generates a list of documents that either have not been published or have been modified and need to be published again or have been deleted since the last publish. The command is found on the **SmartPlant** menu in the Drawings and Reports task.

TIP This command can also be accessed from using the **Find** button on the **Publish** tab of the **Publish** dialog box. For more information, see *Publish Tab (Publish Dialog Box)* (on page 531).

Documents must be up-to-date, and the required **Discipline** property must be defined in order for the documents to be available for publishing. An error message displays if one or more of the documents found by the **Find Documents to Publish** command do not meet this criterion. All items matching the publish criteria continue through the process. For more information on setting the appropriate properties, see *Set properties for publishing documents* (on page 521). For more information on updating documents, see *Updating Documents* (on page 82).

The SmartPlant > Find Documents to Publish command looks for the following:

- Documents created but never published
- Documents modified since their last publish
- Documents deleted after being published

The command looks for documents that need to be republished in the active WBS project. For example, if **Project A** is the active project, the **Find Documents to Publish** command looks for documents in **Project A** only. You set the active WBS project in the **Active Project** box on the main toolbar. For more information, see *Manage Projects* (on page 539).

Find Documents to Publish Dialog Box (on page 538)

Find documents to publish

Documents must be up-to-date, and the required **Discipline** property must be defined in order for the documents to be available for publishing. An error message displays if one or more of the documents found by the **Find Documents to Publish** command do not meet this criterion. For more information on setting the appropriate properties, see *Set properties for publishing documents* (on page 521). For more information on updating documents, see *Updating Documents* (on page 82).

1. From the Drawings and Reports task, click SmartPlant > Find Documents to Publish.

• TIPS

- This command is available only if you have registered the model using the SmartPlant Registration Wizard.
- This feature is also available by clicking the Find button on the Publish dialog box.
- The Find Documents to Publish command determines which documents need to be published or re-published and displays the results of the search in the Find Documents to Publish dialog box.

- From the Select documents to publish list, check the boxes corresponding to the documents that you want to publish.
 - TIP You can quickly select the entire list by clicking **Select All**, or you can clear the entire list by clicking **Clear All**.
- 3. Click **OK** to accept the selections. The documents selected for publishing now display in the **Documents to Publish** list on the **Publish** dialog box and can be saved by publishing the documents. For more information, see *Publish documents* (on page 526).

Find Documents to Publish Dialog Box

Allows you to search for documents that have been updated since they were last published. Additionally, you can use this dialog box to terminate documents that were previously published but no longer exist in the authoring tool. You can access the **Find Documents to Publish** dialog in two ways:

- Select SmartPlant > Find Documents to Publish.
- Click Find on the Publish command dialog box.

Last Published

Displays the date when the files were last searched. The information displaying in the lists on this dialog box was found on this specified date and time. This option is not available in the current release.

Update

Displays the Update dialog box, which allows you to define new search criteria for finding documents to publish. This option is not available in the current release.

Select documents to publish

Displays a list of files that were either updated since they were last published or files that have not yet been published. For each file, this list displays the file name and type, and the date on which the document was last published. If the file has not been published, the **Last Published** field for the document is **New**.

Select documents to terminate

Displays a list of the files that were previously published but have since been removed from the project. For each file, this list box displays the file name and type, and the date on which the document was last published.

Select All

Selects all of the files in the associated list of documents.

Clear All

Clears any selected documents in the associated list.

See Also

Publishing Documents (on page 517)
Find Documents to Publish (on page 536)
Publish Tab (Publish Dialog Box) (on page 531)

Manage Projects

The Work Breakdown Structure (WBS) project is shown in the dropdown at the upper left-hand corner of the Drawings and Reports task window, next to the **Permission Group** dropdown. It shows the current active project.



In the Common task, you can create new WBS items and projects or edit existing ones. For more information, see the *Common User's Guide*.

You use projects in conjunction with publishing. The active project must be set before using the **Final Publish** command on the **SmartPlant** menu.

See Also

Publishing Documents (on page 517)
Select Active Project Dialog Box (on page 539)

Select Active Project Dialog Box

Specifies the active project. You can access the **Select Active Project** dialog box by clicking **More** in the **Active Project** box on the main toolbar. You define whether you want to look in the local **Workspace** or in the **Database** for the project by selecting the options at the top of the dialog box. The project hierarchy updates with the selection of the option.

See Also

Manage Projects (on page 539)

APPENDIX A

Appendix: Troubleshooting Drawings and Reports

This section describes Drawings and Reports error levels and error logging. You can use log files to review activities and errors that occur when working with the software.

Error Levels

Run the [Product Folder]\Core\Tools\Administrator\Bin\EnableErrorLog.exe tool to enable detailed error logging in the SP3DErrors.log. For more information, contact Intergraph Support Services. You can find support information on our web site at http://support.intergraph.com/).

Generally speaking, the larger the number for the error level, the more exhaustive is the logging of errors. The error levels are as follows:

- 1 General user error. This is the default level. At this level, log files only contain error messages for certain anticipated error conditions (such as a missing filter or a missing view style), as well as some unanticipated error messages.
- 101 Development-specific error level. At this level, log files include everything from the previous error level, as well as certain development-specific error or informational messages.
- 201 General Information. At this level, log files include everything from the previous error levels, as well as informational messages about what projects and methods are being called.
- 251 Batch Information. At this level, log files include everything from the previous error levels, as well as special batch-specific informational messages.
- 999 Exhaustive. At this level, log files contain all informational and error messages. When the error level is set to 999, the error log files can become very large.

Log Files

There are three categories of orthographic drawings error log files. These files are all located in your temporary (Temp) folder. For example, your Temp folder might be located at C:\Documents and Settings\login name\Local Settings\Temp.

TIP You can type %Temp% in the Address box at the top of Windows Explorer to go to your Temp location.

The log files are:

- %TEMP%\Drawings.log and %TEMP\Errors.log General purpose error log file. Most of the errors encountered in Drawings and Reports are logged here.
- C:SP3DBatchSvcTemp\Drawings_Batch.log The batch service-specific error log file. Any errors or information messages related to the batch service are found here.
- C:SP3DBatchSvcTemp\DwgBatchServer_< Process ID >_< yyyy_mm_dd_hh_nn_ss >.log The batch server-specific error log files. Any errors or information messages related to the
 batch servers are found here.

- C:SP3DBatchSvcTemp\DwgBatchTier_< Process ID >_< yyyy_mm_dd_hh_nn_ss >.log The batch tier-specific error log files. Any errors or information messages related to the
 batch tier services are found here.
- Generation-time error logs (for example, error log files generated during a drawing update operation). You can easily view these files by right-clicking a drawing in the software and selecting View Log on the shortcut menu.
- TIP The Temp location also includes .xml files. The software creates one .xml file for each graphical view in a drawing as it processes the drawing.

Out of Memory Tips

If you are receiving Out of Memory errors when processing very large drawings, check the following:

- Set the Intersection Edges option in the view style to Off. Sometimes, this option is set to High Resolution for large drawings. High Resolution is typically useful for drawings of one object, such as a hanger drawing or an equipment drawing, where you want to see detail such as the coped intersection of a nozzle with a tank cylinder. If you have this option set high in a large plan drawing, then the software examines every small beam, cylinder, or nut and tries to draw coped intersections or rounded edges in the webs. This operation uses a lot of time and memory.
- Make sure the Preserve Z Order option in the view style is turned off unless you really need it. A case where you might need it is in a cable tray drawing where trays are stacked or crossing in plan. However, for most drawings, this setting just leads to increased processing time and high memory usage for little gain.
- Make sure multiple aspects have not been chosen in the VHL graphic rules.
- If you have a very large session active in the host and then update a drawing, the update process starts at the large memory usage in the active session. So, if you limit your workspace to a very small set of objects or even one object, the update process has a better chance of succeeding. If you use batch updates instead of local updates, workspace is not an issue unless you are using your local computer as the batch server.
- Decrease the Flush Threshold setting (the default is 2000). This setting is available on the Drawing View Properties dialog box for a view in a composed drawing. In some instances, increasing the number may help. During the update process, the Drawings software asks Core for the monikers of all the objects it has to process. Core passes in the monikers and Drawings binds them, thus using memory. The software binds up to 2000 of the objects before it releases the memory and grabs the next 2000. If you reduce this number, say in increments of 500, the memory gets released much more often and can lead to a successful update. It is an iterative process to figure out a good number. This value should be from 5 to 5000 with a default of 2000.
- Set the Geometry Validation setting to ON (the default is OFF). This setting is available on the Drawing View Properties dialog box for a view in a composed drawing or a marine mode drawings-by-rule drawing. If the drawing has SAT or DGN files, there are sometimes many invalid geometry errors that can use increasing amounts of memory. With this setting as On, the software analyzes geometry, then discards and does not draw invalid geometry, thus freeing up resources and reducing update time. The caveat is that the drawing may not be 100% accurate because invalid geometries may represent legitimate items but are removed from the drawing. If the reference file is there just as background, this situation might be acceptable.

- In general, you should not place huge SAT files as equipment shapes. If you must place them as shapes, break them into smaller files.
- The range inside reference files should be kept as small as possible. For example, if the file contains a pipe rack far away from the global origin, place the file in the model and then move it to the appropriate position.

Isometric Drawings

The isometric drawing log files reside at the location specified in your Temp environment variable. For example, the path to the log might be C:\Documents and Settings\login name\Local Settings\Temp.

You can view message files (.mes), piping component files (.pcf), and .xml files for the isometric drawing generation process.

VHL Precision

Objects can be displayed as Vector Hidden Line (VHL). There can be instances when you may want the hidden line information for a model. For example, it may be useful to know how the edges in the hidden line view are occluded. In general, **VHL Precision** should not be set, unless you have some parts in the model that have been modeled with a precision different from rest of the system. The values you can set impact the VHL calculations. The **VHL Precision** setting must be a positive number between 0.001 and 0.000001. This setting is available on the **Drawing View Properties** dialog box for a view in a composed drawing or a marine mode drawings-by-rule drawing.

ISOKEEPFILES Variable

This variable exports XML settings to the location specified in your Temp environment variable. An XML file contains the name of the isometric view style currently being used. This information can be used to troubleshoot isometric drawing settings.

TIP The XML file containing the isometric view style will be approximately 27 KB in size.

The ISOKEEPFILES variable is not a Drawings and Reports setting, it is a System Properties setting.

- 1. Click **Start** and right-click **My Computer**.
- 2. Select Properties.
- 3. Select the Advanced tab.
- 4. Click Environment Variables.

The Temp environment variable is listed under **User variables for** *username*. If you are unsure of where your Temp folder is, the location is noted in this box.

- 5. On the Environment Variables dialogue box, select New under the System variables box.
- 6. Type ISOKEEPFILES in the Variable box and type YES in the Value box.
- 7. Click OK.

NOTE You must set this variable before entering Drawings and Reports and creating the isometric drawings.

Reports

The log file for reports (SP3DReports.log) resides at the location specified in your Temp environment variable. For example, the path to the log might be C:\Documents and Settings\login name\Local Settings\Temp.

See Also

Drawings and Reports (on page 18)
Troubleshooting Linked Servers (on page 543)

Troubleshooting Linked Servers

If your Site/Catalog/Model database server is different from your Reports database server, you can use linked servers for communication between the data sources. However, if linked servers are not configured correctly, the login may fail when you run queries against a linked server.

For linked servers to work correctly, the following must be true:

- The database link must be created on the Site/Catalog/Model database server, not the Reports database server.
- The linked server must support **SQL Server and Windows Authentication Mode**.
- You must be connected to SQL Server using Windows Authentication Mode on both servers.
- Security account delegation must be available on the client and the sending server.

For more information about setting up linked servers, see Microsoft SQL Server documentation.

See Also

Appendix: Troubleshooting Drawings and Reports (on page 540)

APPENDIX B

Appendix: Isometric Drawings and WBS Items

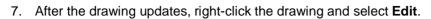
You can document a collection of parts that are assigned to a Work Breakdown Structure (WBS) item with individual isometric drawings.

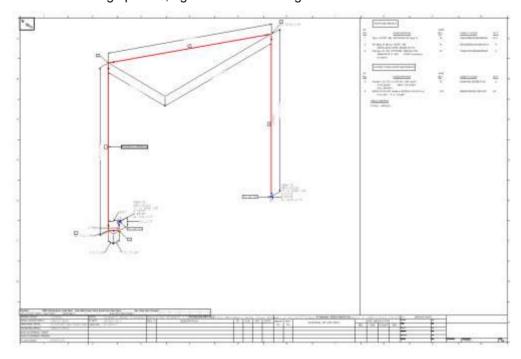
This section describes the workflow for assigning parts to the WBS item and creating isometric drawings from the WBS information.

- 1. In the Piping task, assign the piping objects to an active project with the **Project > Claim** command. For more information on the **Claim** command, see *Claim Objects* in the *Common User's Guide*.
- After assigning objects to an active project, assign them to a WBS item with the Project >
 Assign to WBS command. For more information on the Assign to WBS command, see
 Assign Objects to a WBS Item in the Common User's Guide.
- 3. In the Drawings and Reports task, create a new **Drawings by Query Manager** component with the following options:
 - Select the Catalog Filters > Default Filters > SP3D Object Filters > Object Types > Common > WBS Items > WBS Items Iso Drawing filter
 - Select the WBS Iso Fabrication package.

For more information on creating a **Drawings by Query Manager** component, see *Set up a Drawings by Query Manager component* (on page 110).

- 4. Right-click the Drawings by Query Manager component and select Run Query.
- 5. Right-click the WBS Iso Fabrication component and select Create Drawing(s).
- Right-click the WBS Iso Fabrication component and select Update Now to update the WBS Isometric drawing.





APPENDIX C

Appendix: Personal Isogen Return Values

If an isometric extraction fails, Isogen returns an error number that can be referenced from an error message or the extraction error log within the software. Descriptions for the Isogen error message numbers are shown in the table below.

Error Value	Description
-1001	Failed to create the specified Pre/Post processor.
-1002	Pre/Post processor has crashed.
-1003	Pre/Post processor has timed out, and the external process has been killed.
-1004	Pre/Post processor has timed out, but the external process cannot be terminated (and is therefore still running).
-1	Error in Isogen data. Check message file.
-2	Isogen .dll failed to load (possible installation problem, missing dependent file, etc.) or crash occurred during execution, which has been trapped by the error handler.
-3	Isogen thread filed to finish.Possible execution loop that has been terminated by Personal Isogen.
0	Iso created successfully.
1	Personal Isogen cannot find the isometric directory specified.
2	Invalid Style for this project. The specified Style cannot be found under the specified project.
3	Specified Project not found. The specified project cannot be found.
4	IDFGEN handshake failed. The handshake passed to the idfdll.dll has not been verified.
	NOTE A number of other "unexpected" failures have been found to generate this error.
5	Could not access specified PCF in TEMFILES.
6	ISOGEN DLL failed to establish current directory.

Error Value	Description
7	ISOGEN DLL failed to change directory.
8	The handshake passed to the <i>pisodll.dll</i> has not been verified.
9	Personal Isogen failed to produce a valid drawing. View message file for details as there are many possible causes.
10	ISOGEN DLL failed to restore working directory.
11	IDFGEN DLL failed to establish current directory.
12	IDFGEN DLL failed to change directory.
13	IDFGEN did not complete successfully. View message file for details. One common cause is that the components in the PCF do not form a connected system.
14	IDFGEN DLL failed to restore working directory.
15	IDFGEN could not delete IPISOGEN\PROGRAMS\FOR036.DAT from a previous run.
16	IDFGEN was unable to create ISOGEN.IDX file. Error opening <i>isogen.idx</i> . Error closing <i>isogen.idx</i> .
17	IDFGEN was unable to copy ISOGEN.FLS to the PROGRAMS directory. The following errors may have occurred:
	Error allocating memory to store a line.
	Zero line length encountered.
	Error opening source isogen.fls (in specified project\isotype).
	Error opening target isogen.fls (in pisogen\programs).
	Error closing source isogen.fls (in specified project\isotype).
	Error closing target isogen.fls (in pisogen\programs).
18	No description available from Alias Personal Isogen.
19	Personal Isogen handshake failed. The handshake passed to the <i>pisogen.dll</i> has not been verified.
20	Unknown error in Personal Isogen. An unexpected path through the program has occurred.
21	PISOGEN DLL could not find Current Working Directory. The current directory cannot be identified.
22	PISOGEN DLL could not Change Directory to \PISOGEN\PROGRAMS.

 Isogen failed to write banner to message file. (Probably cannot find message file path). The following errors may have occurred: Error opening isogen.fls. Unable to find MESSAGE line in isogen.fls. 	e
 Unable to find MESSAGE line in isogen.fls. 	
-	
- Francisco de descripción de la constantidad de la	
Error closing isogen.fls.	
 Error opening message file. 	
Error closing message file.	
24 PISOGEN DLL has been unable to restore current working directory.	
The following errors may have occurred:	
Error opening i-gen.fls for reading.	
Error opening isogen.fls for writing.	
Error opening options file specified in i-gen.fls.	
 Error closing options file specified in i-gen.fls. 	
 Error opening i-gen.opl for writing. 	
 Error closing i-gen.opl. 	
Error closing <i>i-gen.fls</i>.Error closing <i>isogen.fls</i>.	
Problem with the specified isometric root directory.	
27 Problem setting the current isometric root directory.	
28 Invalid file extension.	
Not applicable when running Isogen.	
Warning occurred creating a POD (intermediate) file.	
Error occurred creating a POD (intermediate) file. Error occurred during "pass of when the input IDF file is preprocessed.	"
Error occurred creating a POD (intermediate) file. Error occurred during "pass 2 when the POD file is being created from Isogen data.	2"
Failed to load PODGRAPHICS DLL - used in the creation of graphics output file such as DWG and IGR.	es
PODGRAPHICS process failed - IE. a graphics specific process has failed - for example, user requested IGR output but SmartSketch is not installed.	
Not used at this time.	

Error Value	Description
41	Cannot get path to system temp folder.
42	System temp folder does not exist.
43	Unable to set bore units to those specified in the input file.
44	Unable to perform macro substitution in style ISOGEN.FLS.
45	Unable to restore original style ISOGEN.FLS.
46-49	Not used at this time.
50	Project disk is full - unable to create new output files.
51	File containing temp directory is full - unable to process file.
52-999	Not used at this time.
1000	Expected files missing in Style - E.G. no FLS, no options file.
1001	Requested units combination is inconsistent - probably means a combination of metric bore and imperial coordinates.
1002	Invalid drawing format requested.
1003	Problem deleting an Isogen message file.
1004	Line in the options file has invalid format.
1005	Too many switches in the options file.
1006	Too few switches in the options file.
1007	MicroStation design file output requested but MS_EXE environment variable is not defined.
1008	MicroStation design file output requested but MicroStation not installed.
1009	MS_EXE environment variable does not end with a \.
1010	ISOGEN.FLS missing in Style.
1011	Options file missing specified in ISOGEN.FLS is missing.

The following warning messages may also display:

Error Value	Description
1999	Isogen has returned a FAIL for one or more drawings in the set. This means it has been unable to layout the drawing successfully.
2001	Inconsistent units combination (imperial coordinates and metric bores).
2022	IDFGEN has detected a disconnected pipeline that cannot be resolved. This is known as a HARD disconnect. Drawings have still been created that will indicate the cause of the disconnection.
2023	IDFGEN has detected a disconnected pipeline that cannot be resolved, but the disconnected items are not considered to be fundamental to the pipeline. This is known as a FLOATING disconnect. Drawings have still been created that will indicate the cause of the disconnection.
2024	IDFGEN has detected a disconnected pipeline that has been resolved. This is known as a SOFT disconnect. Drawings have still been created that will indicate the cause of the disconnection.

See Also

Edit Options (Isogen Isometric Drawing Component Shortcut Menu) (on page 134)

APPENDIX D

Appendix: Symbols and Symbol Keys

The Drawings and Reports task delivers a symbol library containing Isogen symbols. You can use the Intergraph Symbol Editor to modify the symbols.

Each symbol is associated with a unique code called a symbol key (SKEY). SKEYs contain 2-4 letters; the first two letters define the type of component, and the last two letters define the end type such as flanged, butt welded, or screwed. You can specify a wildcard for the end type. For example, when ** is specified for the end type, the software reads the End Prep from the part and assigns the correct Isogen end type.

You can map the symbol keys using the **Edit Options** command. This command opens **Isogen Configuration**, the tool used to control all the options related to the appearance and information content of the various styles of isometric drawings.

■ NOTES

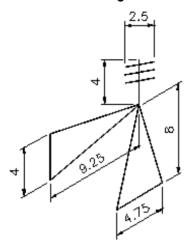
- Before you modify a symbol, review the dimensions and connect points. For more information about the position of SKEY connection points, see *Isogen SKEY Dimensions* (on page 553).
- For a list of delivered piping part classes and their corresponding SKEYs, see Isogen Symbol Key Mapping (on page 601).
- Additional SKEYs are available for use in customizing piping part classes. For more information, see the Alias document titled *Isogen Symbol Key (SKEY) Definitions Reference Guide*, available from Help > Printable Guides. To learn more about Alias, visit their web site (http://www.alias.ltd.uk/) (http://www.alias.ltd.uk/).
- When defining 3- and 4-way valves, only define one leg of the valve. The software will copy the re-defined leg to the remaining legs. You must ensure that the leg of the valve you redefine has the connection to the left and the center point or spindle point to the right.
- The @ character in the symbol keys can be replaced with an integer value in the range 1 to 9, inclusive, to denote the number of segments. Currently, regardless of the value assigned to @, the software draws the symbol per the SKEY plotted isometric shape.
- The + character in the symbol keys can be replaced with an integer value in the range 1 to 9, inclusive, to denote the bend radius.
- You can map most components using an end preparation of 0. However, a notable exception is the nipple. For nipples, you must use two different SKEYs and Piping Component File (PCF) component identifiers, based on the end preparation. In other words, the SKEY mapping for nipples always requires end-prep information.
- If, during drawing extraction, the software encounters a component that lacks SKEY mapping data, the extraction continues as long as the **Drawing.Controls.NoSymbolMapOk** option is set to **True**. The resulting drawing displays the component with an empty gap, along with an error message that points to the gap. In addition, the message file contains an error message.

In This Appendix

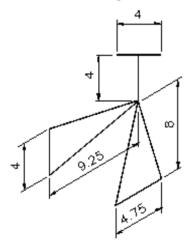
Isogen SKEY Dimensions	553
Isogen Symbol Key Mapping	601
Caps (SKEYs)	644
Couplings (SKEYs)	646
Crosses (SKEYs)	647
Elbows and Bends (SKEYs)	
End Prep Connections	651
Fixed Length Pipes (SKEYs)	651
Flanges (SKEYs)	652
Inline Filters (SKEYs)	
Instruments (SKEYs)	
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Olets (SKEYs)	
Operators (SKEYs)	665
Other End Connections (SKEYs)	667
Reducers (SKEYs)	
Tees (SKEYs)	
Traps (SKEYs)	674
Valves (SKEYs)	
Vents (SKEYs)	678
Welds (SKEYs)	

Isogen SKEY Dimensions

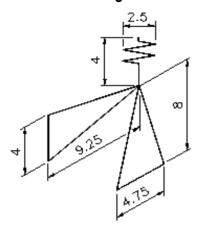
ARFL: Valve - Angle Relief/Vent



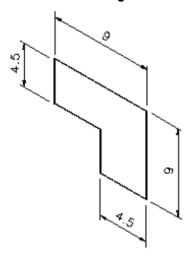
AVFL: Valve - Angle



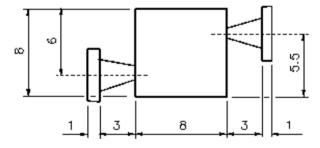
AXFL: Valve - Angle Pressure Reducing



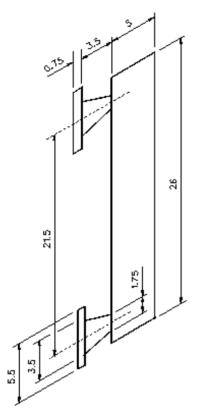
BAFL: Block - Angle



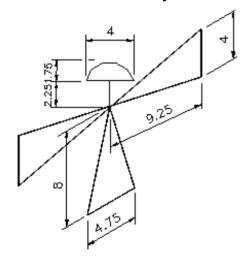
BOFL: Block - Offset



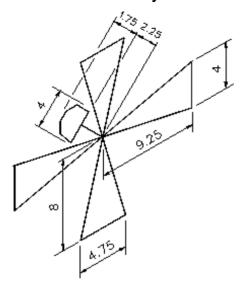
BRFL: Block



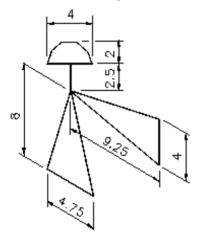
C3FL: Valve - Three-Way Control



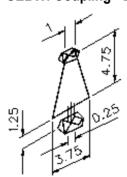
C4FL: Valve - Four-Way Control



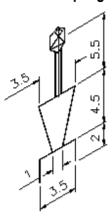
CAFL: Valve - Angle Control



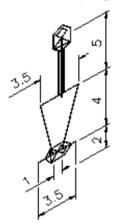
CEBW: Coupling - Elbolet (Butt Weld)



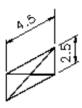
CESC: Coupling - Elbolet (Screwed)



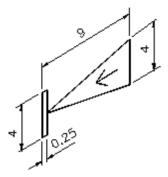
CESW: Coupling - Elbolet (Socket Weld)



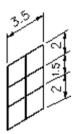
CHFL: Hose Coupling



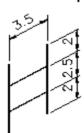
CKFL: Valve - Check



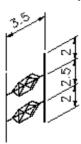
COCP: Coupling - Compression



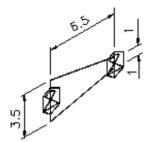
COSC: Coupling - Screwed



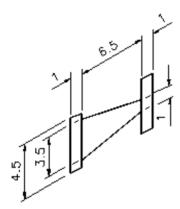
COSW: Coupling - Socket Weld



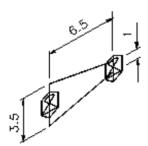
CPBW: Reducer - Concentric Fabricated from Plate



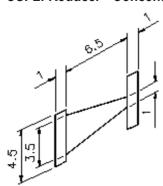
CPFL: Reducer - Concentric Fabricated from Plate (Flanged)



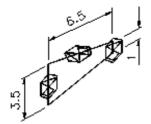
CSBW: Reducer - Concentric Swaged From Plate



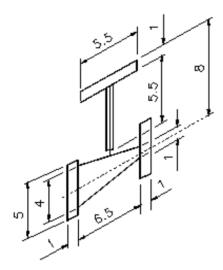
CSFL: Reducer - Concentric Swaged From Plate (Flanged)



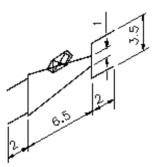
CTBW: Reducer - Concentric with a Connection (Butt Weld)



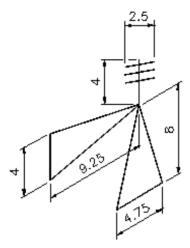
CTFL: Reducer - Concentric with a Connection (Flanged)



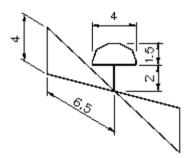
CTSC: Reducer - Concentric with a Connection (Screwed)



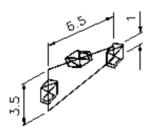
CTSW: Concentric with a Connection (Socket Weld)



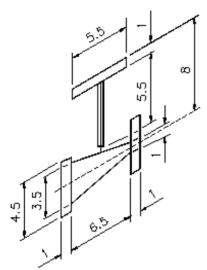
CVFL: Valve - Control



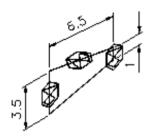
CXBW: Reducer - Concentric with a Connection Swaged from Pipe



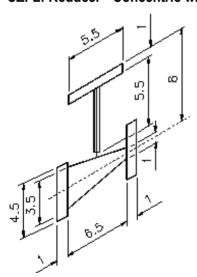
CXFL: Reducer - Concentric with a Connection Swaged from Pipe (Flanged)



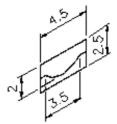
CZBW: Reducer - Concentric with a Connection Fabricated from Plate



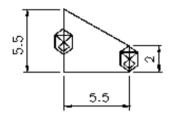
CZFL: Reducer - Concentric with a Connection Fabricated from Plate (Flanged)



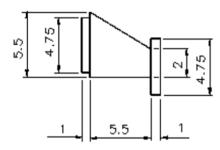
DR: Rupture Disk



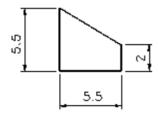
EPBW: Reducer - Eccentric Fabricated from Plate



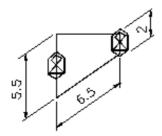
EPFL: Reducer - Eccentric Fabricated from Plate (Flanged)



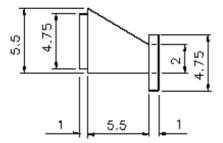
EPSW: Reducer - Eccentric Fabricated from Plate (Socket Weld)



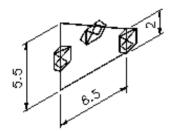
ESBW: Reducer - Eccentric Swaged from Pipe



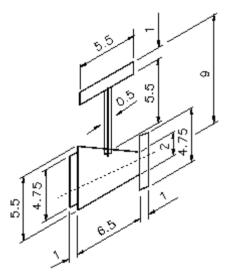
ESFL: Reducer - Eccentric Swaged from Pipe (Flanged)



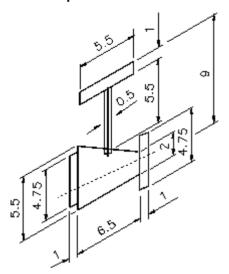
EXBW: Reducer - Eccentric with a Connection Swaged from Pipe



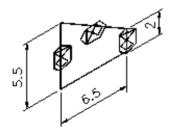
EXFL: Reducer - Eccentric with a Connection Swaged from Pipe (Flanged)



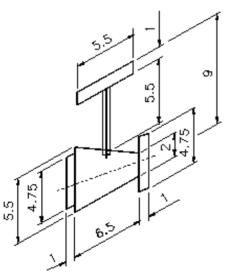
EXFL: Expansion Bellows



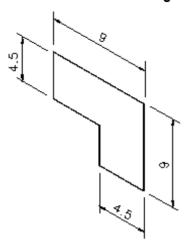
EZBW: Reducer - Eccentric with a Connection Fabricated from Plate



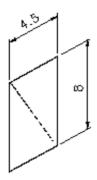
EZFL: Reducer - Eccentric with a Connection Fabricated from Plate (Flanged)



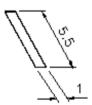
FAFL: Filter/Strainer - Angle



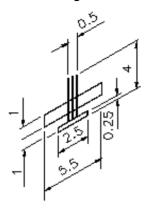
FIFL: Filter/Strainer - Straight Through



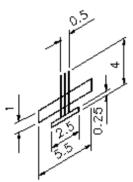
FLBL: Flange - Blind



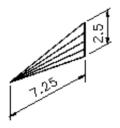
FLFL: Flange - Flared/Loose Backing



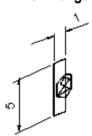
FLLB: Flange - Backing



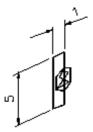
FLOW: Flow Arrow



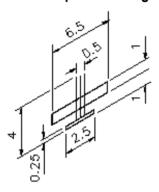
FLRC: Flange - Reducing Concentric



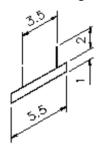
FLRE: Flange - Reducing Eccentric



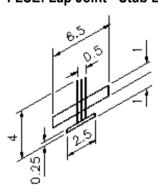
FLRG: Lap Joint - Ring Loose



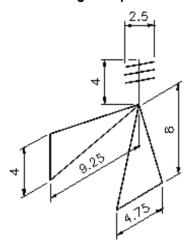
FLSC: Flange - Screwed



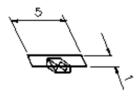
FLSE: Lap Joint - Stub End Loose



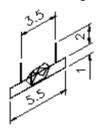
FLSJ: Flange - Slip On with J-type Weld



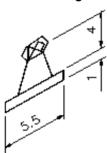
FLSO: Flange - Slip On



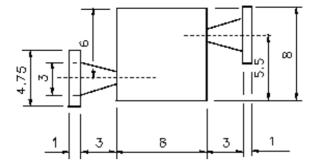
FLSW: Flange - Socket Weld



FLWN: Flange - Weld Neck



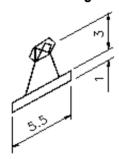
FOFL: Filter/Strainer - Offset



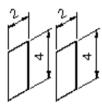
FOSO: Flange - Orifice (Slip On)



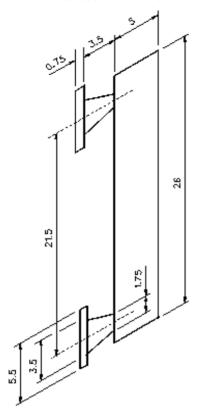
FOWN: Flange - Orifice (Weld Neck)



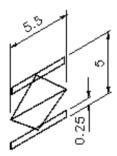
FPPL: Fixed Length Pipe - Without Flanged Ends



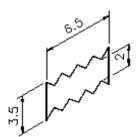
FRFL: Filter/Strainer - Return



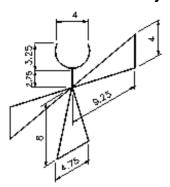
FTFL: Flame Trap



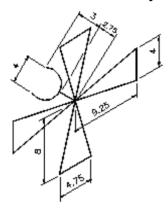
FXFL: Filter/Strainer - Return



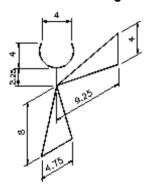
H3FL: Valve - Three-Way Control (Hand Indicator)



H4FL: Valve - Four-Way Control (Hand Indicator)



HAFL: Valve - Angle Control (Hand Indicator)



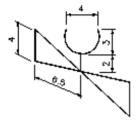
HCSC: Olet - Half Coupling (Screwed)



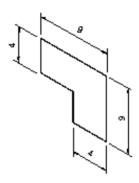
HCSW: Olet - Half Coupling (Socket Weld)



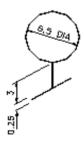
HVFL: Valve - Control Hand Indicator



IAFL: Instrument - Angle



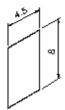
IDFL: Instrument - Dial (Flanged)



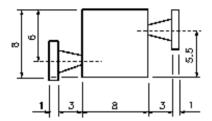
IDPL: Instrument - Dial



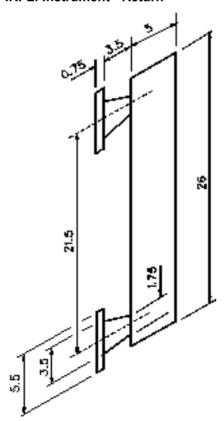
IIFL: Instrument



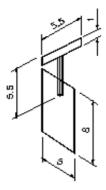
IOFL: Instrument - Offset



IRFL: Instrument - Return



ITFL: Instrument - Tee



KABW: Cap - Butt Weld



KACP: Cap - Compression



KASC: Cap - Screwed



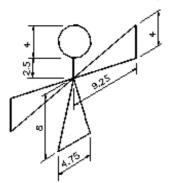
KASW: Cap - Socket Weld



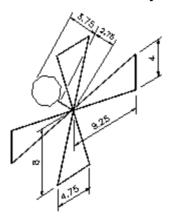
LABW: Olet - Latrolet (Buttweld)



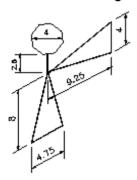
M3FL: Valve - Three-Way Control (Motorized Indicator)



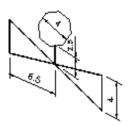
M4FL: Valve - Four-Way Control (Motorized Indicator)



MAFL: Valve - Angle Control (Motorized Indicator)



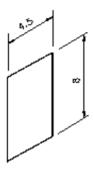
MVFL: Valve - Control (Motorized Indicator)



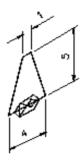
NBSC: Nipple - Screwed



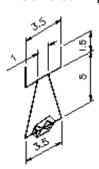
NCFL: Non-Category Item



NIPL: Olet - Nipolet (Plain End)



NISC: Olet - Nipolet (Screwed)



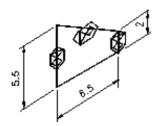
NRSC: Nippled - Screwed



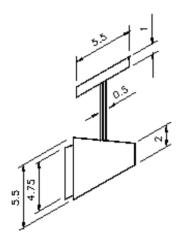
OP: Orifice Plate



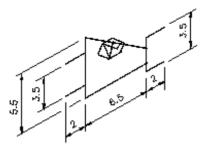
OTBW: Reducer - Eccentric with a Connection (Butt Weld)



OTFL: Reducer - Eccentric with a Connection (Flanged)



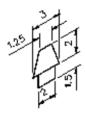
OTSC: Reducer - Eccentric with a Connection (Screwed)



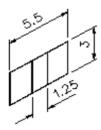
PF: Pipe Block - Fixed Length



PL: Plug



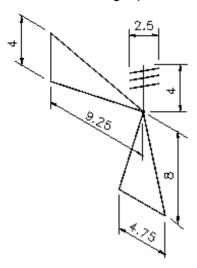
PR: Restrictor Plate



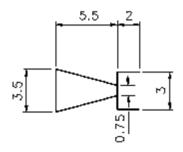
PV: Pipe Block - Variable Length



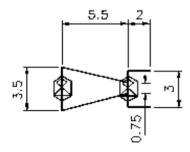
RAFL: Valve - Angle (Relief/Vent)



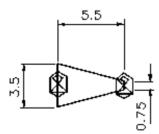
RBSC: Reducer - Concentric (Screwed Bush)



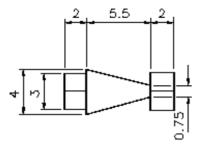
RBSW: Reducer - Concentric (Socket Weld)



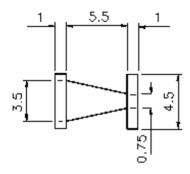
RCBW: Reducer - Concentric (Butt Weld)



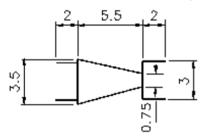
RCCP: Reducer - Concentric (Compression)



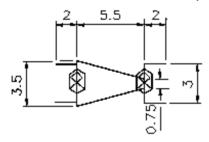
RCFL: Reducer - Concentric (Flanged)



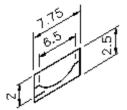
RCSC: Reducer - Concentric (Screwed)



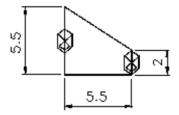
RCSW: Reducer - Concentric (Socket Weld)



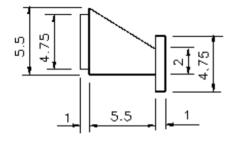
RD: Rupture Disk



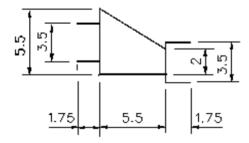
REBW: Reducer - Eccentric (Butt Weld)



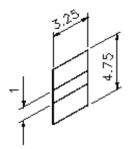
REFL: Reducer - Eccentric (Flanged)



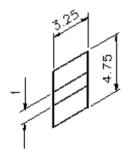
RESC: Reducer - Eccentric (Screwed)



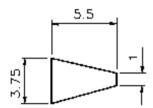
RF: Reducer - Special Reducing Flange



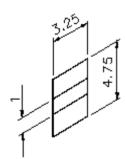
RFPL: Reducer - Connection Block



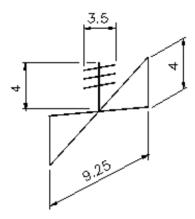
RNSC: Reducer - Concentric (Nipple)



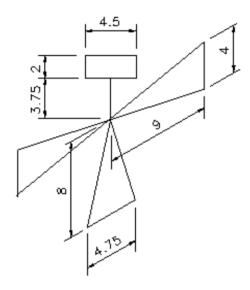
RP: Restrictor Plate



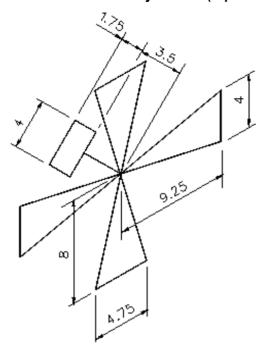
RVFL: Valve - Relief/Vent



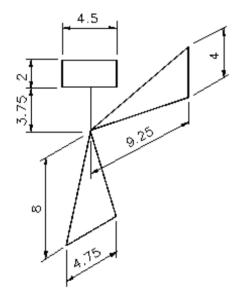
S3FL: Valve - Three-Way Control (Square Indicator)



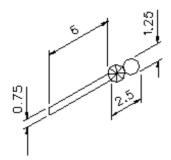
S4FL: Valve - Four-Way Control (Square Indicator)



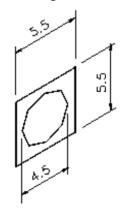
SAFL: Valve - Angle Control (Square Indicator)



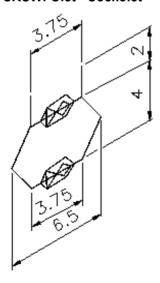
SB: Spectacle - Blind



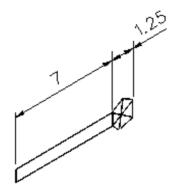
SGFL: Sight Glass



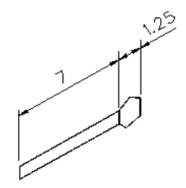
SKSW: Olet - Sockolet



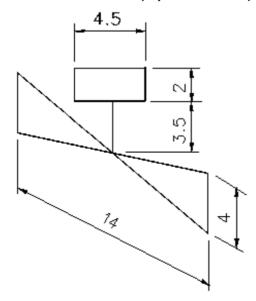
SP: Slip Plate



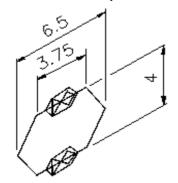
SR: Slip Ring



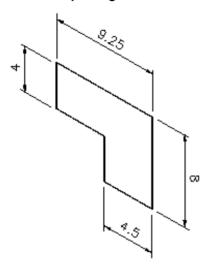
SVFL: Valve - Control (Square Indicator)



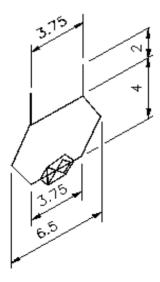
SWBW: Olet Sweepolet



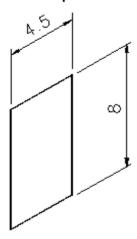
TAFL: Trap - Angle



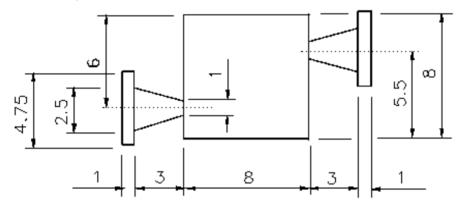
THSC: Olet - Thredolet



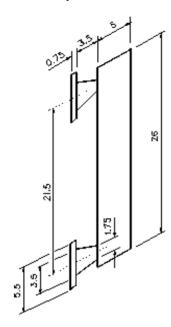
TIFL: Trap - Inline Straight Through



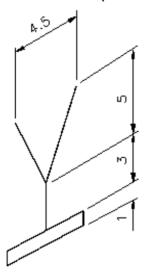
TOFL: Trap - Offset



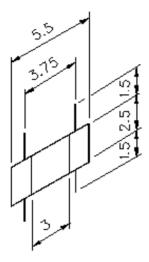
TRFL: Trap - Return



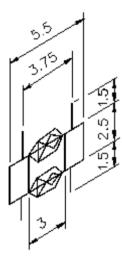
TUFL: Tundish (Funnel)



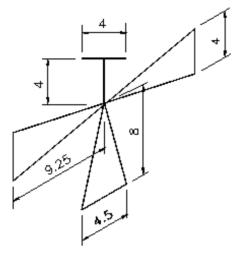
UNSC: Union - Screwed



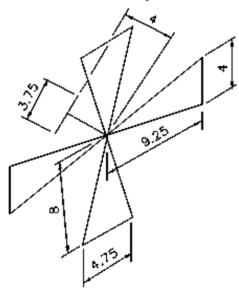
UNSW: Union - Socket Weld



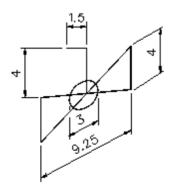
V3FL: Valve - Three-Way



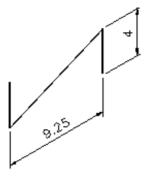
V4FL: Valve - Four-Way



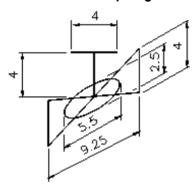
VBFL: Valve - Ball



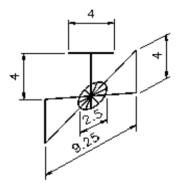
VCFL: Valve - Check



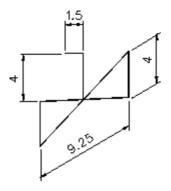
VDFL: Valve - Diaphragm



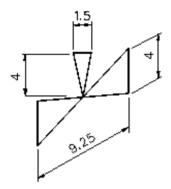
VGFL: Valve - Globe



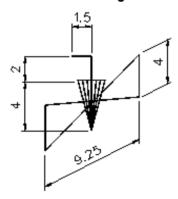
VKFL: Valve - Cock



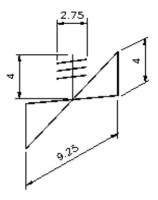
VNFL: Valve - Needle



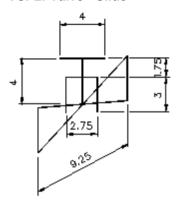
VPFL: Valve - Plug



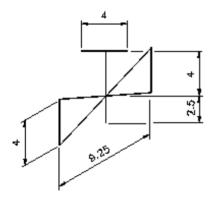
VRFL: Valve - Relief/Vent



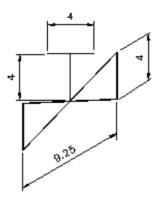
VSFL: Valve - Slide



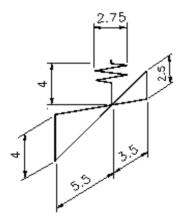
VTFL: Valve - Gate



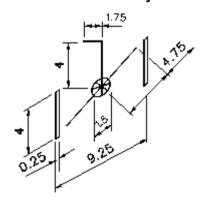
VVFL: Valve - Basic



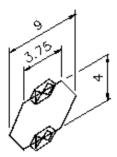
VXFL: Valve - Pressure Reducing



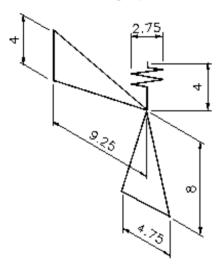
VYFL: Valve - Butterfly



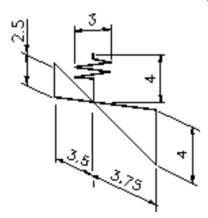
WTBW: Olet - Weldolet



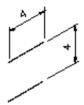
XAFL: Valve - Angle (Pressure Reducing)



XVFL: Valve - Pressure Reducing



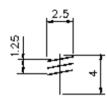
01HG: Support/Hanger



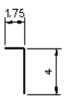
01SP: Used on valves with AV, V3, V4, VD, VG, or VV as the first two characters of their SKEYs



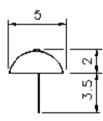
02SP: Used on valves with AR, RA, VR as the first two characters of their SKEYs.



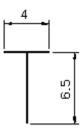
03SP: Used on valves with VB, VK, VY as the first two characters of their SKEYs.



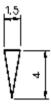
04SP: Not Used



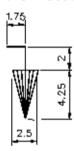
05SP: Used on valves with VT as the first two characters of their SKEYs.



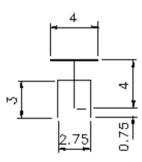
06SP: Used on valves with VN as the first two characters of their SKEYs.



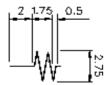
07SP: Used on valves with VP as the first two characters of their SKEYs.



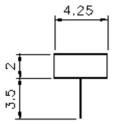
08SP: Used on valves with VS as the first two characters of their SKEYs.



09SP: Used on valves with AX, VX, or XA as the first two characters of their SKEYs.



10SP: Used on valves with SA, SV, S3, or S4 as the first two characters of their SKEYs.



11SP: Used on valves with MA, MV, M3, or M4 as the first two characters of their SKEYs.



See Also

Appendix: Symbols and Symbol Keys (on page 552)

Isogen Symbol Key Mapping

The table below maps which PCF symbol key and PCF component is used for each Smart 3D part class.

NOTE To report an additional port on a piping component as a tap, define the port as a **Tap By System** in the piping component symbol and in the reference data of the piping part. Do not redefine the port if it is an inlet or an outlet.

Smart 3D Part Class	End Prep Code	PCF Skey	PCF Component
11_25DegElbow	0	EL**	ELBOW
11_25DegElbowPDB13	0	EL**	ELBOW
11_25DegElbowPDB163	0	EL**	ELBOW
11_25DegElbowPDB164	0	EL**	ELBOW
11_25DegElbowPDB176	0	EL**	ELBOW
11_25DegElbowPDB177	0	EL**	ELBOW
11_25DegElbowPDB35	0	EL**	ELBOW
11_25DegElbowPDB36	0	EL**	ELBOW
11_25DegElbowPDB704	0	EL**	ELBOW
11_25DegElbowPDB706	0	EL**	ELBOW
11_25DegElbowPDB707	0	EL**	ELBOW
11_25DegElbowPDB708	0	EL**	ELBOW
11_25DegElbowPDB709	0	EL**	ELBOW
11_25DegElbowPDB710	0	EL**	ELBOW
11_25DegElbowPDB711	0	EL**	ELBOW
180DegReturn	0	EU**	ELBOW
1by8BendDoubleYbyFace	0	CR**	CROSS
1by8BendDoubleYbySeat	0	CR**	CROSS
1by8BendDoubleYF12C	0	CR**	CROSS
1by8BendDoubleYTanLen1	0	CR**	CROSS
1by8BendDoubleYTanLen3	0	CR**	CROSS

Smart 3D Part Class	End Prep Code	PCF Skey	PCF Component
1by8BendWyebyFace	0	TY**	TEE
1by8BendWyebySeat	0	TY**	TEE
1by8BendWyeF12C	0	TY**	TEE
1by8BendWyeTanLen1	0	TY**	TEE
1by8BendWyeTanLen3	0	TY**	TEE
1by8ReducingWye	0	TY**	TEE
22_5DegElbow	0	EL**	ELBOW
22o5DegElboPDB13	0	EL**	ELBOW
22o5DegElboPDB160	0	EL**	ELBOW
22o5DegElboPDB161	0	EL**	ELBOW
22o5DegElboPDB162	0	EL**	ELBOW
22o5DegElboPDB163	0	EL**	ELBOW
22o5DegElboPDB164	0	EL**	ELBOW
22o5DegElboPDB165	0	EL**	ELBOW
22o5DegElboPDB170	0	EL**	ELBOW
22o5DegElboPDB175	0	EL**	ELBOW
22o5DegElboPDB176	0	EL**	ELBOW
22o5DegElboPDB177	0	EL**	ELBOW
22o5DegElboPDB179	0	EL**	ELBOW
22o5DegElboPDB180	0	EL**	ELBOW
22o5DegElboPDB185	0	EL**	ELBOW
22o5DegElboPDB36	0	EL**	ELBOW
22o5DegElboPDB691	0	EL**	ELBOW
22o5DegElboPDB692	0	EL**	ELBOW
22o5DegElboPDB694	0	EL**	ELBOW

Smart 3D Part Class	End Prep Code	PCF Skey	PCF Component
22o5DegElboPDB695	0	EL**	ELBOW
22o5DegElboPDB696	0	EL**	ELBOW
22o5DegElboPDB697	0	EL**	ELBOW
22o5DegElboPDB698	0	EL**	ELBOW
22o5DegElboPDB699	0	EL**	ELBOW
22o5DegElboPDB700	0	EL**	ELBOW
22o5DegElboPDB701	0	EL**	ELBOW
22o5DegElbowPDB10	0	EL**	ELBOW
22o5DegElbowPDB35	0	EL**	ELBOW
22o5DegElbowPDB690	0	EL**	ELBOW
22o5DegElbowPDB693	0	EL**	ELBOW
2PRVA1	0	CA**	INSTRUMENT-ANGLE
30DegAsymElbow	301	ELBW	ELBOW
30DegElbow	0	EL**	ELBOW
30DegElbowPDB13	0	EL**	ELBOW
30DegElbowPDB163	0	EL**	ELBOW
30DegElbowPDB164	0	EL**	ELBOW
30DegElbowPDB176	0	EL**	ELBOW
30DegElbowPDB177	0	EL**	ELBOW
30DegElbowPDB35	0	EL**	ELBOW
30DegElbowPDB36	0	EL**	ELBOW
30DegElbowPDB683	0	EL**	ELBOW
30DegElbowPDB684	0	EL**	ELBOW
30DegElbowPDB685	0	EL**	ELBOW
30DegElbowPDB686	0	EL**	ELBOW

Smart 3D Part Class	End Prep Code	PCF Skey	PCF Component
30DegSymElbow	301	ELBW	ELBOW
30P1	0	OP	INSTRUMENT
3WayBall	0	V3**	VALVE-3WAY
3WayBallValve	0	V3**	VALVE-3WAY
3WayBallValvePDB10	0	V3**	VALVE-3WAY
3WayBallValvePDB11	0	V3**	VALVE-3WAY
3WayBallValvePDB13	0	V3**	VALVE-3WAY
3WayCheckValve	0	K3**	VALVE
3WayCheckValve1	0	K3**	VALVE
3WayCheckValve2	0	K3**	VALVE
3WayControlValvePDB10	0	V3**	VALVE-3WAY
3WayControlValvePDB11	0	V3**	VALVE-3WAY
3WayControlValvePDB13	0	V3**	VALVE-3WAY
3WayDiaDiv	21	C3FL	INSTRUMENT-3WAY
3WayDiverterValvePDB10	0	V3**	VALVE-3WAY
3WayDiverterValvePDB11	0	V3**	VALVE-3WAY
3WayDiverterValvePDB13	0	V3**	VALVE-3WAY
3WayGlobeValvePDB10	0	V3**	VALVE-3WAY
3WayGlobeValvePDB11	0	V3**	VALVE-3WAY
3WayGlobeValvePDB13	0	V3**	VALVE-3WAY
3WayPlugValvePDB10	0	V3**	VALVE-3WAY
3WayPlugValvePDB11	0	V3**	VALVE-3WAY
3WayPlugValvePDB13	0	V3**	VALVE-3WAY
45Deg3DElbow	0	EL**	ELBOW
45DegAngleValPDB13	0	AR**	VALVE-ANGLE

Smart 3D Part Class	End Prep Code	PCF Skey	PCF Component
45DegAsymElbow	301	ELBW	ELBOW
45DegElbow	0	EL**	ELBOW
45DegElbowPDB10	0	EL**	ELBOW
45DegElbowPDB13	0	EL**	ELBOW
45DegElbowPDB161	0	EL**	ELBOW
45DegElbowPDB162	0	EL**	ELBOW
45DegElbowPDB163	0	EL**	ELBOW
45DegElbowPDB164	0	EL**	ELBOW
45DegElbowPDB176	0	EL**	ELBOW
45DegElbowPDB177	0	EL**	ELBOW
45DegElbowPDB179	0	EL**	ELBOW
45DegElbowPDB180	0	EL**	ELBOW
45DegElbowPDB35	0	EL**	ELBOW
45DegElbowPDB36	0	EL**	ELBOW
45DegElbowPDB668	0	EL**	ELBOW
45DegElbowPDB669	0	EL**	ELBOW
45DegElbowPDB670	0	EL**	ELBOW
45DegElbowPDB671	0	EL**	ELBOW
45DegElbowPDB672	0	EL**	ELBOW
45DegElbowPDB673	0	EL**	ELBOW
45DegElbowPDB674	0	EL**	ELBOW
45DegElbowPDB675	0	EL**	ELBOW
45DegElbowPDB676	0	EL**	ELBOW
45DegElbowPDB677	0	EL**	ELBOW
45DegElbowPDB678	0	EL**	ELBOW

Smart 3D Part Class	End Prep Code	PCF Skey	PCF Component
45DegElbowPDB679	0	EL**	ELBOW
45DegLRElbow	0	EL**	ELBOW
45DegLTElbow	0	EL**	ELBOW
45DegStreetElbow	0	EL**	ELBOW
45DegStreetElbowFlat	0	EL**	ELBOW
45DegStreetElbowHex	0	EL**	ELBOW
45DegSymElbow	301	ELBW	ELBOW
45DegUnionElbow	0	EL**	ELBOW
4BOX	0	**	INSTRUMENT
4BOX3	0	**	INSTRUMENT
4BOX4	0	**	INSTRUMENT
4CYL	0	**	INSTRUMENT
4CYL3	0	**	INSTRUMENT
4CYL4	0	**	INSTRUMENT
5_625DegElbow	0	EL**	ELBOW
5_625DegElbowPDB13	0	EL**	ELBOW
5_625DegElbowPDB163	0	EL**	ELBOW
5_625DegElbowPDB164	0	EL**	ELBOW
5_625DegElbowPDB176	0	EL**	ELBOW
5_625DegElbowPDB177	0	EL**	ELBOW
5_625DegElbowPDB35	0	EL**	ELBOW
5_625DegElbowPDB36	0	EL**	ELBOW
5_625DegElbowPDB715	0	EL**	ELBOW
5_625DegElbowPDB716	0	EL**	ELBOW
5_625DegElbowPDB717	0	EL**	ELBOW

Smart 3D Part Class	End Prep Code	PCF Skey	PCF Component
5_625DegElbowPDB718	0	EL**	ELBOW
60DegAsymElbow	301	ELBW	ELBOW
60DegElbow	0	EL**	ELBOW
60DegElbowPDB10	0	EL**	ELBOW
60DegElbowPDB13	0	EL**	ELBOW
60DegElbowPDB161	0	EL**	ELBOW
60DegElbowPDB162	0	EL**	ELBOW
60DegElbowPDB163	0	EL**	ELBOW
60DegElbowPDB164	0	EL**	ELBOW
60DegElbowPDB176	0	EL**	ELBOW
60DegElbowPDB177	0	EL**	ELBOW
60DegElbowPDB179	0	EL**	ELBOW
60DegElbowPDB180	0	EL**	ELBOW
60DegElbowPDB35	0	EL**	ELBOW
60DegElbowPDB36	0	EL**	ELBOW
60DegElbowPDB653	0	EL**	ELBOW
60DegElbowPDB654	0	EL**	ELBOW
60DegElbowPDB655	0	EL**	ELBOW
60DegElbowPDB656	0	EL**	ELBOW
60DegElbowPDB657	0	EL**	ELBOW
60DegElbowPDB658	0	EL**	ELBOW
60DegElbowPDB659	0	EL**	ELBOW
60DegElbowPDB660	0	EL**	ELBOW
60DegElbowPDB661	0	EL**	ELBOW
60DegElbowPDB662	0	EL**	ELBOW

Smart 3D Part Class	End Prep Code	PCF Skey	PCF Component
60DegElbowPDB663	0	EL**	ELBOW
60DegElbowPDB664	0	EL**	ELBOW
60DegSymElbow	301	ELBW	ELBOW
72DegElbowPDB10	0	EL**	ELBOW
72DegElbowPDB13	0	EL**	ELBOW
72DegElbowPDB161	0	EL**	ELBOW
72DegElbowPDB162	0	EL**	ELBOW
72DegElbowPDB179	0	EL**	ELBOW
72DegElbowPDB180	0	EL**	ELBOW
72DegElbowPDB35	0	EL**	ELBOW
72DegElbowPDB36	0	EL**	ELBOW
72DegElbowPDB642	0	EL**	ELBOW
72DegElbowPDB643	0	EL**	ELBOW
72DegElbowPDB644	0	EL**	ELBOW
72DegElbowPDB645	0	EL**	ELBOW
72DegElbowPDB646	0	EL**	ELBOW
72DegElbowPDB647	0	EL**	ELBOW
72DegElbowPDB648	0	EL**	ELBOW
72DegElbowPDB649	0	EL**	ELBOW
90Deg3DElbow	0	EL**	ELBOW
90DegAsymElbow	301	ELBW	ELBOW
90DegElbow	0	EL**	ELBOW
90DegLRElbow	0	EL**	ELBOW
90DegLTElbow	0	EL**	ELBOW
90DegRedElbow	421	ERSW	ELBOW-REDUCING

Smart 3D Part Class	End Prep Code	PCF Skey	PCF Component
90DegRedElbowPDB10	0	ER**	ELBOW-REDUCING
90DegRedElbowPDB13	0	ER**	ELBOW-REDUCING
90DegRedElbowPDB161	0	ER**	ELBOW-REDUCING
90DegRedElbowPDB162	0	ER**	ELBOW-REDUCING
90DegRedElbowPDB163	0	ER**	ELBOW-REDUCING
90DegRedElbowPDB164	0	ER**	ELBOW-REDUCING
90DegRedElbowPDB176	0	ER**	ELBOW-REDUCING
90DegRedElbowPDB177	0	ER**	ELBOW-REDUCING
90DegRedElbowPDB179	0	ER**	ELBOW-REDUCING
90DegRedElbowPDB180	0	ER**	ELBOW-REDUCING
90DegRedElbowPDB35	0	ER**	ELBOW-REDUCING
90DegRedElbowPDB36	0	ER**	ELBOW-REDUCING
90DegRedElbowPDB723	0	ER**	ELBOW-REDUCING
90DegRedElbowPDB724	0	ER**	ELBOW-REDUCING
90DegRedElbowPDB725	0	ER**	ELBOW-REDUCING
90DegRedElbowPDB726	0	ER**	ELBOW-REDUCING
90DegRedElbowPDB727	0	ER**	ELBOW-REDUCING
90DegRedElbowPDB728	0	ER**	ELBOW-REDUCING
90DegRedElbowPDB729	0	ER**	ELBOW-REDUCING
90DegRedElbowPDB730	0	ER**	ELBOW-REDUCING
90DegRedStreetElbow	441	ERSC	ELBOW-REDUCING
90DegReducingElbow	0	ER**	ELBOW-REDUCING
90DegReducingStreetElbow	0	ER**	ELBOW-REDUCING
90DegReducingYBranch	0	ER**	ELBOW-REDUCING
90DegSRElbow	0	EL**	ELBOW

Smart 3D Part Class	End Prep Code	PCF Skey	PCF Component
90DegStreetElbow	0	EL**	ELBOW
90DegStreetElbowFlat	0	EL**	ELBOW
90DegStreetElbowHex	0	EL**	ELBOW
90DegSymElbow	301	ELBW	ELBOW
90DegUnionElbow	0	EL**	ELBOW
90DegYBranch	0	EL**	ELBOW
90DoubleElbowPDB10	0	ETCP	ELBOW-TEED
90DoubleElbowPDB13	0	ETCP	ELBOW-TEED
90DoubleElbowPDB35	0	ETCP	ELBOW-TEED
90DoubleElbowPDB36	0	ETCP	ELBOW-TEED
90DoubleElbowTPDB10	0	ET**	ELBOW-TEED
90DoubleElbowTPDB13	0	ET**	ELBOW-TEED
90DoubleElbowTPDB35	0	ET**	ELBOW-TEED
90DoubleElbowTPDB36	0	ET**	ELBOW-TEED
AdapterFlange	211	NCFL	MISC-COMPONENT
ANG	0	CA**	INSTRUMENT-ANGLE
AngleCheckValve	0	AV**	VALVE-ANGLE
AngleControlValve	21	CAFL	INSTRUMENT-ANGLE
AngleGlobeValve	0	AV**	VALVE-ANGLE
AngleGlobeValve_Asym	0	AV**	VALVE-ANGLE
AngleHoseValve	441	AVSW	VALVE-ANGLE
AngleStopCheckValve	0	AV**	VALVE-ANGLE
AngleValve	0	AV**	VALVE-ANGLE
AngleValve45Deg	21	AVFL	VALVE-ANGLE
AngLiftCheckValve	0	AV**	VALVE-ANGLE

Smart 3D Part Class	End Prep Code	PCF Skey	PCF Component
AsymAngleGlobeValve	0	AV**	VALVE-ANGLE
AsymAngleStopCheckVal	0	AV**	VALVE-ANGLE
AsymButterflyValveSym	0	VY**	VALVE
AsymButterflyValveVAL	0	VY**	VALVE
AsymButterflyValveVOH	0	VY**	VALVE
AsymDelugeValve	0	VV**	VALVE
AsymDiaphragmValve150	0	VD**	VALVE
AsymDiaphragmValve300	0	VD**	VALVE
AsymHorLiftCheckValve	0	VC**	VALVE
AsymHorYLiftCheckValve	0	VC**	VALVE
AsymPlugValve	0	VP**	VALVE
AsymSngBasketStrainer	0	FI**	FILTER
AsymStopCheckValve	0	CK**	VALVE
AsymZeroDeadlegTval	0	V3**	VALVE-3WAY
AutoRecirculationValve	0	V3**	VALVE-3WAY
BA3W	0	C3**	INSTRUMENT-3WAY
BallGOpAsymmetrical	0	VB**	VALVE
BallValAsymmetrical	21	VBFL	VALVE
BallValCentertoSeat	421	VBSW	VALVE
BallValEtoEandValFtoF	21	VBFL	VALVE
BallValNipleLenValFtoC	21	VBFL	VALVE
BallValNipleLenValFtoF	21	VBFL	VALVE
BallValSeattoSeat	421	VBSW	VALVE
BallValve	0	VB**	VALVE
BallValve_m	0	VB**	VALVE

Smart 3D Part Class	End Prep Code	PCF Skey	PCF Component
BALR	421	VBSW	VALVE
BALSP	21	VBFL	VALVE
BasinCross	441	CRSC	CROSS
BasketStrainer	0	NC**	MISC-COMPONENT
BasketStrainer2	0	FI**	FILTER
Bellows	0	NC**	MISC-COMPONENT
Bend	421	BESW	BEND
Bend	301	BEBW	BEND
Bend	391	РВ	BEND
Bend	0	BEBW	BEND
BFYHP	211	VYFL	VALVE
BFYHPGO	211	VYFL	VALVE
BlankDisc	121	NCFL	MISC-COMPONENT
BlankGateValAsym	0	VT**	VALVE
BlindFlange	0	FLBL	FLANGE-BLIND
BlindFlange_m	0	FLBL	FLANGE-BLIND
BlindHub	0	NC**	MISC-COMPONENT
BLPAD	0	SP	MISC-COMPONENT
BLSPA	0	SR	MISC-COMPONENT
BLSPO	0	SB	MISC-COMPONENT
BubbleDetector	441	IISC	INSTRUMENT
Bushing	0	UN**	UNION
ButterflyGOpAsym	0	VY**	VALVE
ButterflyValve	0	VY**	VALVE
ButterflyValve_m	121	VYFL	VALVE

Smart 3D Part Class	End Prep Code	PCF Skey	PCF Component
ButterflyValveArmLength	541	VVPF	VALVE
ButterflyvalveElec	201	CVFL	INSTRUMENT
ButterflyValveOnOffV	201	CVFL	INSTRUMENT
ButterflyValveOperHgt	111	VVFL	VALVE
ButterflyValveVarArmLength	0	VY**	VALVE
ButterflyValveVarOpHeigh	0	VY**	VALVE
Сар	0	KA**	CAP
CAPS	0	KA**	CAP
CAPTC	920	KALC	FLANGE-BLIND
CheckValve	0	CK**	VALVE
CheckValve_Fem	0	CK**	VALVE
CheckValveSwing	0	CK**	VALVE
Cl3WayControlVal	0	C3**	VALVE
CIAnalyzer	0	NC**	MISC-COMPONENT
CIAngGlobeControlVal	0	CA**	INSTRUMENT-ANGLE
CIAutoRecirculationVal	0	C3**	INSTRUMENT-3WAY
CIBallControlValve	0	CV**	INSTRUMENT
CIBalloonInstrument	0	IDPL	INSTRUMENT-DIAL
CIDActEPPAngVal	0	IA**	INSTRUMENT-ANGLE
CIDActEPPHWhAngVal	0	IA**	INSTRUMENT-ANGLE
CIDActEPPHWhGISVal	0	CV**	INSTRUMENT
CIDActEPPHWhTopAngVal	0	IA**	INSTRUMENT-ANGLE
CIDActEPPHWhTopGISCVal	0	CV**	INSTRUMENT
CIElecActTy1AngValve	0	IA**	INSTRUMENT-ANGLE
CIElecActTy2AngVal	0	IA**	INSTRUMENT-ANGLE

Smart 3D Part Class	End Prep Code	PCF Skey	PCF Component
CIElecActTy3AngValve	0	IA**	INSTRUMENT-ANGLE
CIElecActTy4AngVal	0	IA**	INSTRUMENT-ANGLE
CIElecActTy5AngValve	0	IA**	INSTRUMENT-ANGLE
CIFlowNozzle	0	NC**	MISC-COMPONENT
CIGateVGearA	0	HV**	INSTRUMENT
CIGlobeControlVal	0	CV**	INSTRUMENT
CILevelIndControlType3	0	II**	INSTRUMENT
CILevelIndControlType4	0	II**	INSTRUMENT
CILevelIndControlType6	0	II**	INSTRUMENT
CIPlugControlValve	0	CV**	INSTRUMENT
CircularSiphonFormD	0	NC**	MISC-COMPONENT
CircularSiphonFormG	0	NC**	MISC-COMPONENT
CIRoDActEPPHWPosC1Val	0	CV**	INSTRUMENT
CIRoDActEPPHWPosD1Val	0	CV**	INSTRUMENT
CISlideControlValve	0	CV**	INSTRUMENT
CITempFusiblePlug	0	II**	INSTRUMENT
CIVacRelRuptureDiscTy1	0	RD	INSTRUMENT
CIVacRelRuptureDiscTy2	0	RD	INSTRUMENT
CIVenturimeter	0	II**	INSTRUMENT
CKL	0	CK**	VALVE
CKLR	421	CKSW	VALVE
CKS	21	CKFL	VALVE
CKWF	121	CKFL	VALVE
Clamp	0	CLMP	CLAMP
ClampHDHWN_TriClover	920	CLMP	CLAMP

Smart 3D Part Class	End Prep Code	PCF Skey	PCF Component
ClamponFlowSensor	301	IIBW	INSTRUMENT
CloseCoupledBranchVal	0	V3**	VALVE-3WAY
CloseCoupledBranchVal2	0	V3**	VALVE-3WAY
CombinationValve	0	V3**	VALVE-3WAY
ConcentricReducer	0	RC**	REDUCER-CONCENTRIC
ConcentricReducer_b	0	RC**	REDUCER-CONCENTRIC
ConcentricReducer_m	0	RC**	REDUCER-CONCENTRIC
ConcentricReducer447_1	0	RC**	REDUCER-CONCENTRIC
ConcentricReducer447_2	0	RC**	REDUCER-CONCENTRIC
ConcentricSwage	0	RC**	REDUCER-CONCENTRIC
ConcReducer	0	RC**	REDUCER-CONCENTRIC
ConductivitySensor	21	IIFL	INSTRUMENT
ConduitGateValve	0	VV**	VALVE
ConservationVent	21	IIFL	INSTRUMENT
ControlGateValve	0	CV**	INSTRUMENT
ControlGateValveOA	0	CV**	INSTRUMENT
Coupling	331	cosc	COUPLING
Coupling	301	CEBW	COUPLING
Coupling	421	COSW	COUPLING
Coupling_m	421	COSW	COUPLING
Coupling_m	331	cosc	COUPLING
Coupling_m	301	CEBW	COUPLING
Coupling_PDB	421	COSW	COUPLING
CPL	421	COSW	COUPLING
CPLB	0	CLMP	CLAMP

Smart 3D Part Class	End Prep Code	PCF Skey	PCF Component
Cross	0	CR**	CROSS
CrossPDB589	0	CR**	CROSS
CrossPDB590_1	0	CR**	CROSS
CrossPDB590_2	0	CR**	CROSS
CrossPDB591	0	CR**	CROSS
CrossPDB592	0	CR**	CROSS
CrossPDB593	0	CR**	CROSS
CrossPDB594	0	CR**	CROSS
CrossPDB595	0	CR**	CROSS
CrossPDB596	0	CR**	CROSS
CrossPDB597	0	CR**	CROSS
CrossPDB598	0	CR**	CROSS
CrossPDB599	0	CR**	CROSS
CrossPDB600	0	CR**	CROSS
CrossPDB601	0	CR**	CROSS
CrossPDB602	0	CR**	CROSS
CrossPDB603	0	CR**	CROSS
CrossPDB604	0	CR**	CROSS
CrossPDB605	0	CR**	CROSS
CrossRed589	0	CR**	CROSS
CrossRed590_1	0	CR**	CROSS
CrossRed590_2	0	CR**	CROSS
CrossRed591	0	CR**	CROSS
CrossRed592	0	CR**	CROSS
CrossRed593	0	CR**	CROSS

Smart 3D Part Class	End Prep Code	PCF Skey	PCF Component
CrossRed597	0	CR**	CROSS
CrossRed598	0	CR**	CROSS
CrossRed599	0	CR**	CROSS
CrossRed600	0	CR**	CROSS
CrossRed601	0	CR**	CROSS
CrossRed602	0	CR**	CROSS
CrossRed603	0	CR**	CROSS
CrossRed604	0	CR**	CROSS
CrossRed605	0	CR**	CROSS
CS3WayBallCVal	0	V3**	VALVE-3WAY
CS3WayControlVal	0	C3**	VALVE-3WAY
CS3WayGlobeCVal	0	C3**	VALVE-3WAY
CS3WayPlugCVal	0	C3**	VALVE-3WAY
CS4WayPlugCVal	0	C4**	VALVE-4WAY
CSAnalyzer	0	NC**	MISC-COMPONENT
CSAngGlobeCVal	0	CA**	VALVE-ANGLE
CSAutoRecircVal	0	C3**	INSTRUMENT-3WAY
CSBallCValve	0	VB**	VALVE
CSBallonInstr	0	IDPL	INSTRUMENT-DIAL
CSBalloonInstr	0	**	INSTRUMENT
CSBasketStrainer	0	NC**	MISC-COMPONENT
CSBoxSinPort	0	NC**	MISC-COMPONENT
CSCylSinPort	0	NC**	MISC-COMPONENT
CSExhaustHead	0	NC**	MISC-COMPONENT
CSExpnJointOff	0	EX**	MISC-COMPONENT

Smart 3D Part Class	End Prep Code	PCF Skey	PCF Component
CSFArrestorTy1	0	FT**	MISC-COMPONENT
CSFilter	0	NC**	MISC-COMPONENT
CSFlowNozzle	0	NC**	MISC-COMPONENT
CSFreeVent	0	NC**	MISC-COMPONENT
CSGlobeCVal	0	VG**	VALVE
CSInlineSilencer	0	NC**	MISC-COMPONENT
CSISteamTrapTy4	0	TO**	TRAP-OFFSET
CSISteamTrapTy5	0	TR**	TRAP-RETURN
CSISteamTrapTy6	0	TO**	TRAP-OFFSET
CSISteamTrapTy7	0	TO**	TRAP-OFFSET
CSISteamTrapTy8	0	TO**	TRAP-OFFSET
CSLIndCtrlTy3	0	**	INSTRUMENT
CSLIndCtrlTy4	0	**	INSTRUMENT
CSLIndCtrlTy6	0	II**	INSTRUMENT
CSPlugCValve	0	VP**	VALVE
CSRDiscVac2	0	RD	SAFETY-DISC
CSRuptureDiscV1	0	RD	SAFETY-DISC
CSSlideCValve	0	VS**	VALVE
CSSprayNozzle	0	NC**	MISC-COMPONENT
CSTFPlug	0	PL**	MISC-COMPONENT
CSTStrainer	21	TEFL	TEE
CSTStrainer	0	TE**	TEE
CSValve	0	VV**	VALVE
CSValveOpGear	0	VV**	VALVE
CSValveOpHandwheel	0	VV**	VALVE

Smart 3D Part Class	End Prep Code	PCF Skey	PCF Component
CSValveOpWrench	0	VV**	VALVE
CSVentSilencer	0	NC**	MISC-COMPONENT
CSVenturimeter	0	II**	INSTRUMENT
CSWOSteamTrap	0	TO**	TRAP-OFFSET
CSYStrainer	0	TE**	TEE
CylinderValve	0	AV**	VALVE-ANGLE
DIA	21	VDFL	VALVE
DIALT	301	VDBW	VALVE
DIAMM	920	VDLC	VALVE
DiaphragmValveBody	0	VD**	VALVE
DiaphragmValveBodyPDB5	0	VV**	VALVE
DIAWM	0	VD**	VALVE
DistancePiece	121	PFFL	PIPE-FIXED
DoublerPlate_1A	0	NC**	MISC-COMPONENT
DoublerPlate_1D	0	NC**	MISC-COMPONENT
DoublerPlate_7	0	NC**	MISC-COMPONENT
DoubleTaperedSpacer1	0	NC**	MISC-COMPONENT
DoubleTaperedSpacer2	0	NC**	MISC-COMPONENT
DoubleTaperedSpacer3	0	NC**	MISC-COMPONENT
DoubleWye	0	CY**	CROSS
DoubleWyebySeat	0	CY**	CROSS
DoubleWyePDB525	0	CY**	CROSS
DoubleWyePDB526	0	CY**	CROSS
DoubleWyePDB527	0	CY**	CROSS
DoubleWyeTanLen1	0	CY**	CROSS

Smart 3D Part Class	End Prep Code	PCF Skey	PCF Component
DresserCoupling	0	NC**	MISC-COMPONENT
DripRingTee	0	IT**	INSTRUMENT-TEE
E225MiterNCut	301	MIBW	BEND
E30MiterNCut	301	MIBW	BEND
E45	0	EL**	ELBOW
E45LR	0	EL**	ELBOW
E45LRRJ	301	ELBW	ELBOW
E45LT	301	ELBW	ELBOW
E45LTType1	0	EL**	ELBOW
E45MiterNCut	301	MIBW	BEND
E45TC	920	ELLC	ELBOW
E60MiterNCut	301	MIBW	BEND
E90	0	EL**	ELBOW
E90LR	0	EL**	ELBOW
E90LRRJ	301	ELBW	ELBOW
E90LRT	391	ETLR	ELBOW-TEED
E90LRT	301	ETLR	ELBOW-TEED
E90LT	301	ELBW	ELBOW
E90LTType1	0	EL**	ELBOW
E90MiterNCut	301	MIBW	BEND
E90SR	301	ELBW	ELBOW
E90TC	920	ELLC	ELBOW
EccentricReducer	0	RE**	REDUCER-ECCENTRIC
EccentricReducer_b	0	RE**	REDUCER-ECCENTRIC
EccentricReducer_m	301	REBW	REDUCER-ECCENTRIC

Smart 3D Part Class	End Prep Code	PCF Skey	PCF Component
EccentricSwage	0	RE**	REDUCER-ECCENTRIC
Elbolet	0	CE**	COUPLING
Elbolet_m	0	CE**	COUPLING
Elbow45Deg	421	ELSW	ELBOW
Elbow45Deg	301	ELBW	ELBOW
Elbow45Deg	331	EBSC	ELBOW
Elbow45Deg	441	ELSC	ELBOW
Elbow45Deg	11	ELFL	ELBOW
Elbow45Deg	391	ELBW	ELBOW
Elbow45Deg	21	ELFL	ELBOW
Elbow90Deg	441	ELSC	ELBOW
Elbow90Deg	421	ELSW	ELBOW
Elbow90Deg	331	EBSC	ELBOW
Elbow90Deg	301	ELBW	ELBOW
Elbow90Deg	11	ELFL	ELBOW
Elbow90Deg	391	ELBW	ELBOW
Elbow90Deg	21	ELFL	ELBOW
ElevatedFireMonitorTy1	11	NCFL	MISC-COMPONENT
ElevatedFireMonitorTy2	11	NCFL	MISC-COMPONENT
Endolet	0	HC**	OLET
Endolet_m	0	HC**	OLET
EqualSteamTrap	0	TO**	TRAP-OFFSET
EqualSteamTrap3	0	TO**	TRAP-OFFSET
ExhaustHead	441	NCSC	MISC-COMPONENT
ExpnJoint	11	NCFL	MISC-COMPONENT

Smart 3D Part Class	End Prep Code	PCF Skey	PCF Component
ExpnJointOffset	21	NCFL	MISC-COMPONENT
FBLD	0	FLBL	FLANGE-BLIND
FemaleAdapter	0	NC**	MISC-COMPONENT
FemaleAdapterSizeChange	0	NC**	MISC-COMPONENT
FemaleHoseCoupling	0	CH**	MISC-COMPONENT
FemaleToKwikFl	0	NC**	MISC-COMPONENT
FERR	920	FLFL	FLANGE
FERR	301	FEBW	FLANGE
Filter	21	NCFL	MISC-COMPONENT
FireHydrant	11	NCFL	MISC-COMPONENT
FittingReducer	0	RC**	REDUCER-CONCENTRIC
FlameArrestor	21	NCFL	MISC-COMPONENT
FlangeAdapter	0	NC**	MISC-COMPONENT
FlangeConnectionPiece	0	FLWN	FLANGE
FlangedElbowPipet	0	CEBW	ELBOLET
FlangOlet	0	WTFL	OLET
FlexCoupling	0	NC**	MISC-COMPONENT
FlexibleCoupling	0	NC**	MISC-COMPONENT
FloatTrapType1	0	TI**	TRAP
FloatTrapType2	0	TI**	TRAP
FloatTrapType3	0	TR**	TRAP-RETURN
FloatTrapType4	0	TI**	TRAP
FloatTrapType5	0	TI**	TRAP
FloatTrapType6	0	TR**	TRAP-RETURN
FlowMagnetic	21	**	INSTRUMENT

End Prep Code	PCF Skey	PCF Component
331	**	INSTRUMENT
21	**	INSTRUMENT
21	VVFL	VALVE
11	NCFL	MISC-COMPONENT
21	FLRC	FLANGE-REDUCING- CONCENTRIC
0	FLRC	FLANGE-REDUCING- CONCENTRIC
0	FLSW	FLANGE
0	FLWN	FLANGE
0	VV**	VALVE
21	VTFL	VALVE
421	VTSW	VALVE
21	VTFL	VALVE
21	VTFL	VALVE
21	VTFL	VALVE
421	VTSW	VALVE
0	VV**	VALVE
0	VT**	INSTRUMENT
0	VV**	VALVE
421	VTSW	VALVE
21	VTFL	VALVE
21	VTFL	VALVE
	331 21 21 21 11 21 0 0 0 0 0 21 421 21 421 0 0 0 0 0 0 0 0 0 0 0 0 0 1	21

Smart 3D Part Class	End Prep Code	PCF Skey	PCF Component
GATR	421	VTSW	VALVE
GATRASYM	0	VT**	VALVE
GaugeRootValSingleOut	0	VV**	VALVE
GaugeRootValSingleOutR	0	VV**	VALVE
GaugeValMultiOut	0	V4**	VALVE-4WAY
GaugeValMultiOutR	0	V4**	VALVE-4WAY
GLO	0	VG**	VALVE
GlobeControlValve	21	CV**	INSTRUMENT
GlobeValAsymmetrical	21	VGFL	VALVE
GlobeValCentertoSeat	421	VGSW	VALVE
GlobeValEtoEandValFtoF	21	VGSW	VALVE
GlobeValNipleLenValFtoC	21	VGSW	VALVE
GlobeValNipleLenValFtoF	21	VGSW	VALVE
GlobeValSeattoSeat	421	VGSW	VALVE
GlobeValve	0	VG**	VALVE
GlobeValve_Asym	0	VG**	VALVE
GlobeValve_m	0	VG**	VALVE
GlobeValveMotor	0	VG**	VALVE
GLOR	421	VGSW	VALVE
HalfCoupling	0	HC**	OLET
HalfCouplingEtoE	0	HCSW	OLET
HalfCouplingFtoF	0	HCSW	OLET
HalfCouplingFtoHC	0	HCSW	OLET
HMI1	0	II**	INSTRUMENT
HorYLiftCheckValveFtoC	0	VC**	VALVE

Smart 3D Part Class	End Prep Code	PCF Skey	PCF Component
HorYLiftCheckValveFtoF	0	VC**	VALVE
HoseConnection	0	CH**	MISC-COMPONENT
HoseRackStation	441	NCSC	MISC-COMPONENT
Hub	0	NC**	MISC-COMPONENT
HubAndSpigotReducer636	0	RC**	REDUCER-CONCENTRIC
I2WAY	0	CV**	INSTRUMENT
I3WAY	0	C3**	INSTRUMENT-3WAY
IA1	0	IDPL	INSTRUMENT-DIAL
IA2	0	IDPL	INSTRUMENT-DIAL
IADD	0	II**	INSTRUMENT
ICM1	0	II**	INSTRUMENT
ICM2	0	II**	INSTRUMENT
ICM3	0	II**	INSTRUMENT
ICM4	0	II**	INSTRUMENT
ICM5	0	II**	INSTRUMENT
IDCO	0	NC**	INSTRUMENT
IDL1C	0	CV**	INSTRUMENT
IDL1D	0	CV**	INSTRUMENT
IDL4C	0	CV**	INSTRUMENT
IDL4D	0	CV**	INSTRUMENT
IDR1A	0	CV**	INSTRUMENT
IDR1B	0	CV**	INSTRUMENT
IDR4A	0	CV**	INSTRUMENT
IDR4B	0	CV**	INSTRUMENT
IEAM1	0	MV**	INSTRUMENT

Smart 3D Part Class	End Prep Code	PCF Skey	PCF Component
IEAM2	0	MV**	INSTRUMENT
IEAM3	0	MV**	INSTRUMENT
IEAM4	0	MV**	INSTRUMENT
IEAM5	0	MV**	INSTRUMENT
IIOL	0	OP	INSTRUMENT
IIOR	0	OP	INSTRUMENT
IIV	0	**	INSTRUMENT
IKG1	0	SV**	INSTRUMENT
IKG2	0	SV**	INSTRUMENT
ILSV	0	II**	INSTRUMENT
IMDC2	0	NC**	INSTRUMENT
IMFM1	0	II**	INSTRUMENT
IMFM2	0	**	INSTRUMENT
IMFM3	0	**	INSTRUMENT
IMFM4	0	II**	INSTRUMENT
IMOP1	0	OP	INSTRUMENT
IMRSV	0	II**	INSTRUMENT
IncreasingTee	0	TE**	TEE
IND	0	II**	INSTRUMENT
InlineSilencerTy1	21	NCFL	MISC-COMPONENT
InsertPlate	0	NC**	MISC-COMPONENT
InstIndicator	0	IDPL	INSTRUMENT-DIAL
InstrumentTee	920	ITFL	INSTRUMENT-TEE
InstrumentTee	301	ITFL	INSTRUMENT-TEE
InvertedCheckValve	441	VRSC	VALVE

Smart 3D Part Class	End Prep Code	PCF Skey	PCF Component
InvrtdBucktStmTrapTy1	0	TI**	TRAP
InvrtdBucktStmTrapTy2	0	TO**	TRAP-OFFSET
InvrtdBucktStmTrapTy3	0	TI**	TRAP
InvrtdBucktStmTrapTy4	0	TI**	TRAP
InvrtdBucktStmTrapTy5	0	TI**	TRAP
InvrtdBucktStmTrapTy6	0	TI**	TRAP
IOP	0	OP	INSTRUMENT
IPDA1	0	SV**	INSTRUMENT
IPDA2	0	SV**	INSTRUMENT
IPDA3	0	SV**	INSTRUMENT
IPDA3A	0	SV**	INSTRUMENT
IPDA4	0	SV**	INSTRUMENT
IPDA4A	0	SV**	INSTRUMENT
IPDA5	0	SV**	INSTRUMENT
IPL1C	0	SV**	INSTRUMENT
IPL1D	0	SV**	INSTRUMENT
IPL4C	0	SV**	INSTRUMENT
IPL4D	0	SV**	INSTRUMENT
IPR1A	0	SV**	INSTRUMENT
IPR1B	0	SV**	INSTRUMENT
IPR4A	0	SV**	INSTRUMENT
IPR4B	0	SV**	INSTRUMENT
IPR4D	0	SV**	INSTRUMENT
IPSR1	0	SV**	INSTRUMENT
IPSR2	0	SV**	INSTRUMENT

Smart 3D Part Class	End Prep Code	PCF Skey	PCF Component
IPSR3	0	SV**	INSTRUMENT
IPSR4	0	SV**	INSTRUMENT
IR1	0	IR**	INSTRUMENT-RETURN
IR2	0	**	INSTRUMENT
IR3	0	IR**	INSTRUMENT-RETURN
IR4	0	IR**	INSTRUMENT-RETURN
IR5	0	**	INSTRUMENT
IR6	0	**	INSTRUMENT
IR7	0	**	INSTRUMENT
IR8	0	II**	INSTRUMENT
IR9	0	IR**	INSTRUMENT-RETURN
IREG1	0	CV**	INSTRUMENT
IREG2	0	CV**	INSTRUMENT
IREG3	0	CV**	INSTRUMENT
IREG4	0	CV**	INSTRUMENT
IREG5	0	CV**	INSTRUMENT
IRI2	0	CV**	INSTRUMENT
IRVT1	0	RA**	INSTRUMENT-ANGLE
IRVT2	0	RA**	INSTRUMENT-ANGLE
IRVT3	0	RA**	INSTRUMENT-ANGLE
IRVT4	0	RA**	INSTRUMENT-ANGLE
IRVT5	0	RA**	INSTRUMENT-ANGLE
IRVT6	0	RA**	INSTRUMENT-ANGLE
IRVT7	0	RA**	INSTRUMENT-ANGLE
IRXPA	0	SV**	INSTRUMENT

Smart 3D Part Class	End Prep Code	PCF Skey	PCF Component
ISCV3	0	CV**	INSTRUMENT
ISSD3	0	C3**	INSTRUMENT-3WAY
ISSDA	0	CA**	INSTRUMENT-ANGLE
ISSDS	0	CV**	INSTRUMENT
ISSPA	0	SA**	INSTRUMENT-ANGLE
ISSPS	0	SV**	INSTRUMENT
ITM	0	**	INSTRUMENT
IVM1	0	**	INSTRUMENT
IVM2	0	II**	INSTRUMENT
IVM3	0	II**	INSTRUMENT
IVM4	0	**	INSTRUMENT
IWFE	0	NC**	INSTRUMENT
KnifeGateValve	0	VS**	VALVE
KnifeGateValvewSM	0	VS**	VALVE
KwikandLargeFl	0	FLWN	FLANGE
LapJointFlange	0	FLSE	FLANGE
LapJointFlange_m	0	FBSE	FLANGE
LAT	21	TEFL	MISC-COMPONENT
Lateral	0	TE**	TEE
Lateral45	0	TE**	TEE
LateralPDB608	0	TE**	TEE
LateralPDB609	0	TE**	TEE
LateralPDB610	0	TE**	TEE
Latrolet	301	LABW	OLET
LevelcontrolvalPDB740	0	VV**	VALVE

Smart 3D Part Class	End Prep Code	PCF Skey	PCF Component
LevelcontrolvalPDB741	0	VV**	VALVE
LevelcontrolvalPDB742	0	VV**	VALVE
LOL	0	LABW	OLET
LongWeldNeckFlange	0	FLWN	FLANGE
MaleAdapter	0	NC**	MISC-COMPONENT
MaleAdapterSizeChange	0	NC**	MISC-COMPONENT
MaleHoseCoupling	0	CH**	MISC-COMPONENT
MaleToKwikFl	0	NC**	MISC-COMPONENT
MassFlow	331	II**	INSTRUMENT
MechanicalJtReducer632	0	RC**	REDUCER-CONCENTRIC
MinKnifeGateValve	0	VS**	VALVE
Monitor	11	NCFL	MISC-COMPONENT
MonitorType1	0	NC**	MISC-COMPONENT
MotOperatedVal	0	MV**	INSTRUMENT
MSCBA3W	0	C3**	INSTRUMENT-3WAY
MultiportDia2WayG	920	V3LC	VALVE-3WAY
MultiportDia2WayOptions	0	V3**	VALVE-3WAY
MultiportDia3Way	920	V4LC	VALVE-4WAY
MultiportDia3WayOptions	0	V4**	VALVE-4WAY
NEE	421	VNSW	VALVE
NIP	301	FPPL	PIPE-FIXED
NIP	331	NRSC	COUPLING
NIP	391	FPPL	PIPE-FIXED
Nipolet	0	NIPL	OLET
Nipple	301	FPPL	PIPE-FIXED

Smart 3D Part Class	End Prep Code	PCF Skey	PCF Component
Nipple	391	FPPL	PIPE-FIXED
Nipple	331	NRSC	COUPLING
NippoletTrim	301	NIBW	OLET
OletGTrim	301	WTBW	OLET
OletTrim	301	WTBW	OLET
OnBranchUnionTee	0	TE**	TEE
OneQuarterBendPDB538	0	EL**	ELBOW
OneQuarterBendPDB539	0	EL**	ELBOW
OneQuarterBendPDB540	0	EL**	ELBOW
OneQuarterBendPDB541	0	EL**	ELBOW
OneQuarterBendPDB542	0	EL**	ELBOW
OneQuarterBendPDB543	0	EL**	ELBOW
OneQuarterBendPDB544	0	EL**	ELBOW
OneQuarterBendPDB545	0	EL**	ELBOW
OneQuarterBendPDB546	0	EL**	ELBOW
OneQuarterBendPDB547	0	EL**	ELBOW
OneQuarterBendPDB548	0	EL**	ELBOW
OneQuarterBendPDB549	0	EL**	ELBOW
OneQuarterBendPDB550	0	EL**	ELBOW
OneQuarterBendPDB551	0	EL**	ELBOW
OneQuarterBendPDB552	0	EL**	ELBOW
OneQuarterBendPDB553	0	EL**	ELBOW
OneQuarterBendPDB554	0	EL**	ELBOW
OneQuarterBendPDB555	0	EL**	ELBOW
OneQuarterBendPDB556	0	EL**	ELBOW

Smart 3D Part Class	End Prep Code	PCF Skey	PCF Component
OneQuarterBendPDB557	0	EL**	ELBOW
OneQuarterBendPDB558	0	EL**	ELBOW
OneQuarterBendPDB559	0	EL**	ELBOW
OneQuarterBendPDB560	0	EL**	ELBOW
OneQuarterBendPDB561	0	EL**	ELBOW
OneQuarterBendPDB562	0	EL**	ELBOW
OneQuarterBendPDB563	0	EL**	ELBOW
OneQuarterBendPDB564	0	EL**	ELBOW
OneQuarterBendPDB565	0	EL**	ELBOW
OneQuarterBendPDB566	0	EL**	ELBOW
OneQuarterBendPDB567	0	EL**	ELBOW
OneQuarterBendPDB568	0	EL**	ELBOW
OneQuarterBendPDB569	0	EL**	ELBOW
OneQuarterBendPDB570	0	EL**	ELBOW
OneQuarterBendPDB571	0	EL**	ELBOW
OneQuarterBendPDB572	0	EL**	ELBOW
OneQuarterBendPDB573	0	EL**	ELBOW
OneQuarterBendPDB574	0	EL**	ELBOW
OneQuarterBendPDB575	0	EL**	ELBOW
OneQuarterBendPDB576	0	EL**	ELBOW
OneQuarterBendPDB577	0	EL**	ELBOW
OneQuarterBendPDB578	0	EL**	ELBOW
OneQuarterBendPDB579	0	EL**	ELBOW
OneQuarterBendPDB580	0	EL**	ELBOW
OneQuarterBendPDB581	0	EL**	ELBOW

Smart 3D Part Class	End Prep Code	PCF Skey	PCF Component
OneQuarterBendPDB582	0	EL**	ELBOW
OneQuarterBendPDB583	0	EL**	ELBOW
OnRunUnionTee	0	TE**	TEE
OrificeFlange	0	FOWN	FLANGE
OrificeFlangeTwoTaps	0	FOWN	FLANGE
OrificePlates	0	OP	INSTRUMENT
Paddle	0	NC**	MISC-COMPONENT
PinchValve	0	VV**	VALVE
PipeSleeve	0	NC**	MISC-COMPONENT
PipeSleeve_3A	0	NC**	MISC-COMPONENT
PipeSleeve_3B	0	NC**	MISC-COMPONENT
PipeSleeveWithSeal	0	NC**	MISC-COMPONENT
PipeSlvWithPuddleFlg	301	NCBW	MISC-COMPONENT
PlainHub	0	NC**	MISC-COMPONENT
PlanarUSiphon	0	NC**	MISC-COMPONENT
PlateFlange	0	FL**	FLANGE
Plug	331	PL	MISC-COMPONENT
PlugGOpAsymmetrical	0	VP**	VALVE
PlugValve	0	VP**	VALVE
PressureGauge	331	IDPL	INSTRUMENT
PressureReducingValve	21	VXFL	VALVE
PressureRegulator	441	II**	INSTRUMENT
PressureReliefValve	0	RA**	INSTRUMENT
PressureReliefValveASym	0	AR**	VALVE-ANGLE
PressureReliefValveSym	21	ARFL	VALVE-ANGLE

Smart 3D Part Class	End Prep Code	PCF Skey	PCF Component
PressureSwitch	441	**	INSTRUMENT
PressureTransmitter	441	II**	INSTRUMENT
PrRedValveWithStrainr	21	VXFL	VALVE
PrRedValveYType	21	VXFL	VALVE
REDC	0	RC**	REDUCER-CONCENTRIC
REDCLT	301	RCBW	REDUCER-CONCENTRIC
REDCMM	920	RCLC	REDUCER-CONCENTRIC
RedCoupling	0	COVT	COUPLING
REDCRJ	301	RCBW	REDUCER-CONCENTRIC
REDE	0	RE**	REDUCER-ECCENTRIC
REDELT	301	REBW	REDUCER-ECCENTRIC
REDEMM	920	RELC	REDUCER-ECCENTRIC
REDERJ	301	REBW	REDUCER-ECCENTRIC
RedLateral	0	TE**	TEE
RedLateral45	0	TE**	TEE
ReducingBranchCross	0	CR**	CROSS
ReducingControlGateValve	0	CV**	INSTRUMENT
ReducingCoupling	421	COSW	COUPLING
ReducingFlange	0	FLRC	FLANGE-REDUCING- CONCENTRIC
ReducingGateValve	21	IIFL	INSTRUMENT
ReducingInsert	0	NC**	MISC-COMPONENT
ReducingInstrumentTee	0	TE**	TEE
ReducingLateral	0	TE**	TEE

Smart 3D Part Class	End Prep Code	PCF Skey	PCF Component
ReducingRunAndBranchCross	0	CR**	CROSS
ReducingRunAndBranchLateral	0	TE**	TEE
ReducingRunAndBranchTee	0	TE**	TEE
ReducingRunBrnchCross	0	CR**	CROSS
ReducingRunBrnchLateral	21	TEFL	TEE
ReducingTee	0	TE**	TEE
ReducingTee_m	0	TE**	TEE
ReducingTeeWye	0	TY**	TEE
ReinforcingPad	0	n/a	REINFORCEMENT-PAD
ReinforcingPad_m	0	n/a	REINFORCEMENT-PAD
ReinforcingPadNonRadial	0	n/a	REINFORCEMENT-PAD
RoseBox	0	NC**	MISC-COMPONENT
RPAD	0	n/a	REINFORCEMENT-PAD
Saddle	0	WTBW	OLET
SanitaryCheckValve	0	VC**	VALVE
SanitaryIncreaser625	0	RC**	REDUCER-CONCENTRIC
SanitaryIncreaser626	0	RC**	REDUCER-CONCENTRIC
SanitaryIncreaser628	0	RC**	REDUCER-CONCENTRIC
SanitaryIncreaser630	0	RC**	REDUCER-CONCENTRIC
SanitaryReducer620	0	RC**	REDUCER-CONCENTRIC
SanitaryReducer621	0	RC**	REDUCER-CONCENTRIC
SanitaryReducer623	0	RC**	REDUCER-CONCENTRIC
SDG1	0	VD**	VALVE
SDR1	0	VD**	VALVE
ShortReducer634	0	RC**	REDUCER-CONCENTRIC

Smart 3D Part Class	End Prep Code	PCF Skey	PCF Component
ShStRotameter	21	IIFL	INSTRUMENT
SingleBasketStrainer	21	FIFL	MISC-COMPONENT
SingleBasketStrainer	11	FIFL	FILTER
SingleTaperedSpacer1	0	NC**	MISC-COMPONENT
SingleTaperedSpacer2	0	NC**	MISC-COMPONENT
SingleTaperedSpacer3	0	NC**	MISC-COMPONENT
SlipOnFlange	0	FLSO	FLANGE
SlipOnFlange_m	0	FLSO	FLANGE
SlipOnReducingFlange	0	FLRC	FLANGE-REDUCING- CONCENTRIC
SMISC1	0	**	INSTRUMENT
SMR1	0	**	INSTRUMENT
SocketFlange	0	FLSW	FLANGE
SocketWeldedFlange	0	FLSW	FLANGE
SocketweldFlange	0	FLSW	FLANGE
SocketWeldReducingFlange	0	FLRC	FLANGE-REDUCING- CONCENTRIC
Sockolet	0	SKSW	OLET
Sockolet_m	0	SKSW	OLET
SOL	0	SKSW	OLET
SolenoidValve	421	CVSW	INSTRUMENT
SOP1	0	OP	INSTRUMENT
SP1	0	CV**	INSTRUMENT
SPACER	0	SR	MISC-COMPONENT
SpectacleBlind	0	SB	MISC-COMPONENT
SpectacleBlind1	0	SB	MISC-COMPONENT

Smart 3D Part Class	End Prep Code	PCF Skey	PCF Component
SPR1	0	CV**	INSTRUMENT
Sprinkler	331	NCSC	MISC-COMPONENT
SREG1	0	**	INSTRUMENT
SRV1	0	CV**	INSTRUMENT
StandardCircularSiphon	0	NC**	MISC-COMPONENT
SteamTrap	0	TOSC	TRAP-OFFSET
SteamTrap3	441	TOSC	TRAP-OFFSET
SteamTrapTy4	441	TOSC	MISC-COMPONENT
SteamTrapTy5	441	TOSC	MISC-COMPONENT
SteamTrapTy6	0	TO**	MISC-COMPONENT
SteamTrapTy7	441	TOSC	MISC-COMPONENT
SteamTrapTy8	11	TOFL	MISC-COMPONENT
StopCheckValve	0	VG**	VALVE
StreetTee	0	TE**	TEE
StuddedCross	31	CRFL	CROSS
StuddedTee	31	TEFL	TEE
SumpStrainer	441	FISC	MISC-COMPONENT
SUPPORT	64	01HG	SUPPORT
SUPPORT	8	01HG	SUPPORT
SUPPORT	95	01HG	SUPPORT
SUPPORT	72	01HG	SUPPORT
SUPPORT	74	01HG	SUPPORT
SUPPORT	12	01HG	SUPPORT
SUPPORT	60	01HG	SUPPORT
SUPPORT	21	01HG	SUPPORT

Smart 3D Part Class	End Prep Code	PCF Skey	PCF Component
SUPPORT	54	01HG	SUPPORT
SUPPORT	30	01HG	SUPPORT
SUPPORT	44	01HG	SUPPORT
SUPPORT	70	01HG	SUPPORT
SUPPORT	43	01HG	SUPPORT
SUPPORT	15	01HG	SUPPORT
SUPPORT	42	01HG	SUPPORT
SUPPORT	31	01HG	SUPPORT
SUPPORT	41	01HG	SUPPORT
SUPPORT	22	01HG	SUPPORT
SUPPORT	91	01HG	SUPPORT
SUPPORT	90	01HG	SUPPORT
SUPPORT	11	01HG	SUPPORT
Sweepolet	0	SWBW	OLET
SweepOletTrim	0	SWBW	OLET
SWGC	391	CSBW	REDUCER-CONCENTRIC
SWGE	391	ESBW	REDUCER-ECCENTRIC
SwingCheckValve	0	VC**	VALVE
Т	0	TE**	TEE
TankBottomValve	0	VC**	VALVE
TankDrainValve	21	VVFL	VALVE
TankVentCheckValve	11	VRFL	VALVE
TappedWyeF2F	0	TY**	TEE
TappedWyeF2S	0	TY**	TEE
TechlokClamp	930	CLMP	CLAMP

Smart 3D Part Class	End Prep Code	PCF Skey	PCF Component
Tee	0	TE**	TEE
Tee_m	0	TE**	TEE
Tee_PDB	21	TEFL	TEE
TeeStrainer	21	FIFL	MISC-COMPONENT
TeeWye	0	TY**	TEE
TeeWyeReducingRRB	0	TY**	TEE
TeeWyeReducingRun	0	TY**	TEE
TempBasketStrainer	111	FIFL	MISC-COMPONENT
TempBasketStrainer1	111	FIFL	MISC-COMPONENT
TempConeStrainer	111	FIFL	MISC-COMPONENT
TemperatureControlValve	441	**	INSTRUMENT
TemperatureSwitch	331	**	INSTRUMENT
TemperatureTransmitter	331	**	INSTRUMENT
TFI	0	IT**	TEE
ThermmoDyTrapType1	0	TI**	TRAP
ThermmoDyTrapType2	0	TI**	TRAP
ThermmoDyTrapType3	0	TI**	TRAP
ThermmoDyTrapType4	0	TI**	TRAP
ThermoStmTrapTy1	0	TA**	TRAP-ANGLE
ThermoStmTrapTy2	0	TI**	TRAP
ThermoStmTrapTy3	0	TI**	TRAP
ThermoStmTrapTy4	0	TI**	TRAP
ThermoStmTrapTy5	0	TI**	TRAP
ThermoStmTrapTy6	0	TA**	TRAP-ANGLE
ThermoStmTrapTy7	0	TI**	TRAP

Smart 3D Part Class	End Prep Code	PCF Skey	PCF Component
ThreadedFlange	0	FLSC	FLANGE
Thredolet	0	THSC	OLET
Thredolet_m	0	THSC	OLET
TLT	301	TEBW	TEE
TMMM	920	TELC	TEE
TMMMS	920	TELC	TEE
TMSWMS	301	TEBW	TEE
TMSWMS	920	TELC	TEE
TOL	0	THSC	OLET
TRB	0	TE**	TEE
TRI	0	IT**	TEE
TrimmableReducingTee	421	TESW	TEE
TrimmableTee	421	TESW	TEE
TrimSymElbowType1	301	ELBW	ELBOW
TrimSymElbowType2	301	ELBW	ELBOW
TRJ	301	TEBW	TEE
TRLT	301	TEBW	TEE
TRMMM	920	TELC	TEE
TRMMMS	920	TELC	TEE
TRUBWWM	0	BU**	MISC-COMPONENT
TrueY	0	TY**	TEE
TrueY2FC	21	TYFL	TEE
TrueY3FC	21	TYFL	TEE
TRUSWWM	0	BU**	MISC-COMPONENT
TRWWM	0	TE**	TEE

Smart 3D Part Class	End Prep Code	PCF Skey	PCF Component
TRWWMS	0	TE**	TEE
TStrainer	0	FI**	MISC-COMPONENT
TWMSW	0	TE**	TEE
TWMW	0	TE**	TEE
TWWM	0	TE**	TEE
TWWMS	301	TEBW	TEE
UN	421	UNSW	UNION
Union	0	UN**	UNION
UnionHead	0	UN**	UNION
UnionTail	0	UN**	UNION
UprightWye	0	TY**	TEE
UprightWyeF32C	0	TY**	TEE
USiphon	0	NC**	MISC-COMPONENT
VE45	0	EL**	ELBOW
VE90Elbow	0	EL**	ELBOW
VentSilencer2	21	NCFL	MISC-COMPONENT
VentValve	0	VR**	VALVE
VentValvePDB508	0	VR**	VALVE
VentValvePDB509	0	VR**	VALVE
VictaulicClamp	0	EL**	CLAMP
VictaulicT	0	EL**	TEE
VortexFlow	21	IIFL	INSTRUMENT
VPSCoupling	915	COVT	COUPLING
VPSElbow45Deg	915	ELLC	ELBOW
VPSElbow90Deg	915	ELLC	ELBOW

Smart 3D Part Class	End Prep Code	PCF Skey	PCF Component
VPSTee	915	TELC	TEE
VREDC	1005	RCCL	REDUCER-CONCENTRIC
VREDE	1005	RECL	REDUCER-ECCENTRIC
WaferCheckValve	121	CKFL	VALVE
WaterFilterPorts90deg	21	FAFL	FILTER
WaterFilterPortsInline	21	FIFL	FILTER
Weld	35	WS	Field Weld at Assembly Site, including Offshore
Weld	40	WF	Field Fit Weld at Assembly Site
Weld	5	WW	Shop Weld
Weld	20	WM	Miter Weld
Weld	30	WS	Field Weld at Assembly Site
Weld	45	WF	Field Fit Weld at Job Site, Including Offshore
Weld	10	WW	Shop Weld at Job Site Fabrication Shop
Weld	25	WM	MiterWeld At Job Site Fabrication Shop
Weld	15	WWA	Automatic Shop Weld
Weld	50	WFT	Field Weld at Job Site for Loose Flange
WeldNeckExpanderFlange	0	FLWN	FLANGE
WeldNeckFlange	0	FLWN	FLANGE
WeldNeckFlange	0	FLWN	FLANGE
WeldNeckFlange_m	0	FLWN	FLANGE
WeldNeckFlangeNonCir	0	FLWN	FLANGE
Weldolet	0	WTBW	OLET

Smart 3D Part Class	End Prep Code	PCF Skey	PCF Component
Weldolet_m	0	WTBW	OLET
WeldoletNonRadial	0	BWTT	OLET
WOL	0	WTBW	OLET
WOSteamTrap	441	TOSC	TRAP-OFFSET
WOSteamTrap3	0	TO**	TRAP-OFFSET
WOSteamTrapOA	0	TO**	MISC-COMPONENT
WyeStrainer	0	FI**	MISC-COMPONENT
WyeStrainer	0	FI**	FILTER
WyeStrainer2	441	FISC	MISC-COMPONENT
WyeStrainer3	441	FISC	MISC-COMPONENT
XLT	301	CRBW	CROSS
XMMMM	920	CRLC	CROSS
XRLT	301	CRBW	CROSS
XRMMMM	920	CRLC	CROSS
Υ	0	TY**	Y-PIECE-FITTING
YStrainer	0	TY**	TEE
YStrainer2	0	FI**	MISC-COMPONENT
ZeroDeadlegTval	0	V3**	VALVE-3WAY
ZeroDeadlegTval2	0	V3**	VALVE-3WAY

Caps (SKEYs)

Description	Key	Plotted Isometric Shape	User-Definable (Yes/No)
Cap - Butt Weld	KABW	0-	Υ
Cap - Compression	KACP	J	Υ
Cap - Screwed	KASC	E	Υ
Cap - Socket Weld	KASW	E	Υ

See Also

Appendix: Symbols and Symbol Keys (on page 552)

Couplings (SKEYs)

Description	Key	Plotted Isometric Shape	User-Definable (Yes/No)
Nipple - Screwed (Running)	NRSC		Y
Nipple - Screwed (Barrel)	NBSC		Y
Coupling - Compression	COCP		Y
Coupling - Screwed	COSC	I	Υ
Coupling - Socket Weld	COSW		Y
Elbolet Coupling Butt Weld	CEBW		Y
Elbolet Coupling Screwed	CESC		Y
Elbolet Coupling Socket Weld	CESW		Υ

See Also

Appendix: Symbols and Symbol Keys (on page 552)

Crosses (SKEYs)

Description	Key	Plotted Isometric Shape	User-Definable (Yes/No)
Cross - Butt Weld	CRBW	+	N
Cross - Compression	CRCP	 -	N
Cross - Flanged	CRFL		N
Cross - Screwed	CRSC) 	N
Cross - Set On	CRSO	\times	N
Cross - Set On Reinforced	CRRF	Reinforced	N
Cross - Socket Weld	CRSW	3 ` €	N

See Also

Appendix: Symbols and Symbol Keys (on page 552)

Elbows and Bends (SKEYs)

The @ character in the symbol keys can be replaced with an integer value in the range 1 to 9, inclusive, to denote the number of segments. Currently, regardless of the value assigned to @, the symbol is drawn per the SKEY plotted isometric shape. The + character in the symbol keys may be replaced with an integer value in the range 1 to 9, inclusive, to denote the bend radius.

Description	Key	Plotted Isometric Shape	User- Definable (Yes/No)
Elbow - Butt Weld (90 and 45)	ELBW	-	N
Elbow - Butt Weld (90 and 45) With A Connection	ETBW	- France	N
Elbow - Butt Weld 180 Return (U Elbow)	EUBW		N
Elbow - Compression (90 and 45)	ELCP	\$ SS	N
Elbow - Compression (90 and 45) With A Connection	ETCP		N
Elbow - Screwed (90 and 45) With Male Ends	EBSC	4"	N
Elbow - Screwed (90 and 45) With Female Ends	ELSC	The second second	N

Description	Key	Plotted Isometric Shape	User- Definable (Yes/No)
Elbow - Screwed - Female (90 and 45) With A Connection	ETSC	TE	N
Elbow - Socket Weld	ELSW	N C	N
Elbow - Socket Weld With A Connection	ETSW	300	N
Bend - Flanged (All Angles)	BEFL	1	N
Bend - Flanged (All Angles) With A Connection	BTFL	7	N
Bend - Flanged 180 Return (U Bend)	BUFL		N
Bend - Miter Flanged	MIFL	· ·	N
Bend - Miter Flanged With A Connection	MTFL		N

Description	Key	Plotted Isometric Shape	User- Definable (Yes/No)
Bend - Miter Butt Weld	MIBW	-	N
Bend - Miter Butt Weld With A Connection	MTBW	- Para	Y
Bend - Lobster Back Flanged	L@FL	1	N
Bend - Lobster Back Flanged With A Connection	T@FL		N
Bend - Lobster Back Butt Weld	L@BW	-	N
Bend - Lobster Back Butt Weld With A Connection	T@BW	- France	N
Bend - Pulled (All Angles)	PB+D		N
Bend - Pulled 180 Return (U Bend)	BU+D		N
Bend - Pulled (All Angles) With A Connection	TB+D		N

End Prep Connections

You can replace the ** characters in the symbol keys with one of the following end prep types:

End Type	End Prep Codelist Value Range	End Prep Type (**)
Flanged	2-199	FL (Flanged)
Male	321 - 329	LC (Liner - clamped)
	331 - 339	SC (Screwed)
	341 - 349	LC (Liner - clamped)
	351 - 359	LC (Liner - clamped)
	361 - 369	CP (Compression)
	371 - 379	SC (Screwed)
	381 - 389	SC (Screwed)
Female	420 - 429	SW (Socket Weld)
	440 - 449	SC (Screwed)
All Other Codelist Values		BW (Butt Weld)

See Also

Fixed Length Pipes (SKEYs)

Description	Key	Plotted Isometric Shape	User- Definable (Yes/No)
Fixed Length Pipe - With Flanged Ends	FPFL		N
Fixed Length Pipe - Without Flanged Ends	FPPL		Υ

See Also

Flanges (SKEYs)

Description	Key	Plotted Isometric Shape	User- Definable (Yes/No)
Flange - Blind (Blank)	FLBL		Υ
Flange - Flared/Loose Backing	FLFL	H	Υ
Flange - Loose Backing	FLLB	H	Υ
Flange - Reducing Concentric	FLRC		Υ
Flange - Reducing Eccentric	FLRE		Υ
Flange - Slip On	FLSO		Υ
Flange - Slip On With 'J' Type Weld	FLSJ		Υ
Flange - Orifice Slip On	FOSO		Υ
Flange - Socket Weld	FLSW		Υ
Flange - Weld Neck	FLWN		Υ

Description	Key	Plotted Isometric Shape	User- Definable (Yes/No)
Flange - Orifice Weld Neck	FOWN		Υ
Jacket Flange - Weld Neck	JFWN		Υ
Jacket Flange - Slip On	JFSO	•	Υ

Inline Filters (SKEYs)

You can replace the ** characters in the symbol keys with an end prep type. For more information, see *End Prep Connections* (on page 651).

Description	Key	Plotted Isometric Shape	User- Definable (Yes/No)
Filter/Strainer - Straight Through	FI**		Y
Filter/Strainer - Angle	FA**	4	N
Filter/Strainer - Offset	FO**		N
Filter/Strainer - Return	FR**		N

See Also

Instruments (SKEYs)

You can replace the ** characters in the symbol keys with an end prep type. For more information, see *End Prep Connections* (on page 651).

Description	Key	Plotted Isometric Shape	User-Definable (Yes/No)
Instrument	**		Y
Instrument - Angle	IA**	4	N
Instrument - Offset	IO**	211MM OFFSET NORTH 114MM OFFSET DOWN	N
Instrument - Return	IR**		N
Instrument - Dial	IDPL		N
Instrument - Dial Flanged	IDFL	DIAL FACE EAST	N
Orifice Plate	OP		Y

Description	Key	Plotted Isometric Shape	User-Definable (Yes/No)
Restrictor Plate	PR		Υ
Rupture Disk	DR		Y
Valve - Angle Relief/Vent	RA**	***	Υ
Valve - Angle Pressure Reducing	XA**		Υ
Valve - Control	CV**	7	Υ
Valve - Angle Control	CA**		Υ
Valve - 3-Way Control	C3**	1	N
Valve - 4-Way Control	C4**		N

Description	Key	Plotted Isometric Shape	User-Definable (Yes/No)
Valve - Control Square Indicator	SV**		Y
Valve - Angle Control Square Indicator	SA**		Y
Valve - 3-Way Control Square Indicator	S3**	The state of the s	N
Valve - 4-Way Control Square Indicator	S4**		N
Valve - Control Motorized Indicator	MV**	2	Y
Valve - Angle Control Motorized Indicator	MA**		Y
Valve - 3-Way Control Motorized Indicator	M3**		N
Valve - 4-Way Control Motorized Indicator	M4**	X	N

Description	Key	Plotted Isometric Shape	User-Definable (Yes/No)
Valve - Control Hand Indicator	HV**		Υ
Valve - Angle Control Hand Indicator	HA**		Y
Valve - 3-Way Control Hand Indicator	H3**	A STATE OF THE STA	N
Valve - 4-Way Hand Indicator	H4**		N
Valve - Pressure Reducing	XV**		Y
Valve - Relief/Vent	RV**		Y

LJSE Type Flanges (SKEYs)

Description	Key	Plotted Isometric Shape	User- Definable (Yes/No)
Lap Joint Ring (Loose Backing Flange)	FLRG	#	Υ
Lap Joint Stub End (Loose Backing Flange)	FLSE		Υ

See Also

Miscellaneous Items (SKEYs)

Description	Key	Plotted Isometric Shape	User- Definable (Yes/No)
Flow Indicator	FLOW		Υ
Hanger/Support	01HG	//	Υ

See Also

Miscellaneous Pipe Components (SKEYs)

You can replace the ** characters in the symbol keys with an end prep type. For more information, see *End Prep Connections* (on page 651).

Description	Key	Plotted Isometric Shape	User- Definable (Yes/No)
Block - Angle	BA**	4	N
Expansion Bellows	EX**		Y
Flame Trap	FT**		Y
Flexible Hose	FX**		Υ
Hose Coupling	CH**		Y
Non-Category Item	NC**		Υ
Block Offset	BO**		N

Description	Key	Plotted Isometric Shape	User- Definable (Yes/No)
Plug	PL		Y
Restrictor Plate	RP		Y
Block - Return	BR**	1	N
Sight Glass	SG**		Y
Slip Plate	SP		Y
Slip Ring	SR		Υ
Spectacle Blind	SB		Y
Tundish (Funnel)	TU**	1	Y

Olets (SKEYs)

Description	Key	Plotted Isometric Shape	User-Definable (Yes/No)
Olet - Half Coupling Screwed	HCSC		Y
Olet - Half Coupling Socket Weld	HCSW		Y
Olet - Latrolet Butt Weld	LABW		Y
Olet - Latrolet Screwed	LASC		Y
Olet - Latrolet Socket Weld	LASW		Υ
Olet - Nipolet Screwed	NISC		Υ
Olet - Nipolet Plain End	NIPL		Υ

Description	Key	Plotted Isometric Shape	User-Definable (Yes/No)
Olet - Sockolet	SKSW		Υ
Olet - Sweepolet	SWBW		Υ
Olet - Thredolet	THSC		Y
Olet - Weldolet	WTBW		Υ
Instrument Tee - Flanged	ITFL		N

Operators (SKEYs)

Description	Key	Plotted Isometric Shape	User- Definable (Yes/No)
Fittings used on valves with AV, V3, V4, VD, VG, and VV as the first two characters of the SKEY	01SP		Υ
Fittings used on valves with AR, RA, and VR as the first two characters of the SKEY	02SP	#	Υ
Fittings used on valves with VB, VK, and VY as the first two characters of the SKEY	03SP		Y
Fittings used on valves with VT as the first two characters of the SKEY	05SP		Υ
Fittings used on valves with VT as the first two characters of the SKEY	06SP		Y
Fittings used on valves with VP as the first two characters of the SKEY	07SP		Y
Fittings used on valves with VS as the first two characters of the SKEY	08SP		Y
Fittings used on control valves with AX , VX , and XA as the first two characters of the SKEY	09SP		Y
Fittings used on control valves with SA , SV , S3 , and S4 as the first two characters of the SKEY	10SP		Y

Description	Key	Plotted Isometric Shape	User- Definable (Yes/No)
Fittings used on control valves with MA , MV , M3 , and M4 as the first two characters of the SKEY	11SP		Υ
Fittings used on control valves with HA , HV , H3 , and H4 as the first two characters of the SKEY	12SP		Υ
Fittings used on control valves with CA , CV , C3 , and C4 as the first two characters of the SKEY	13SP		Y

Other End Connections (SKEYs)

Description	Key	Plotted Isometric Shape	User- Definable (Yes/No)
Erection Weld or Connection on fittings with an SW end	SW	E	Υ
Erection Weld or Connection on fittings with an SC end	SC	С	Υ
Erection Weld or Connection on fittings with a CP end	СР]	Υ

NOTE For SW, SC, or CP type end connections, the software fills in the plotted shapes for erection items and leaves the shapes open for fabrication type items. If the connecting pipe fabrication category is erection, then the software overrides whatever is defined in the fitting.

See Also

Reducers (SKEYs)

Description	Key	Plotted Isometric Shape	User-Definable (Yes/No)
Reducer - Concentric Butt Weld	RCBW	1	Υ
Reducer - Concentric Fabricated From Plate	CPBW	1	Y
Reducer - Concentric Swaged From Pipe	CSBW	1	Y
Reducer - Concentric Butt Weld With a Connection	CTBW		Υ
Reducer - Concentric Fabricated From Plate With a Connection	CZBW		Y
Reducer - Concentric Swaged From Pipe With a Connection	CXBW		Y
Reducer - Concentric Compression	RCCP	- €[⊃=-	Υ
Reducer - Concentric Flanged	RCFL		Y
Reducer - Concentric Fabricated From Plate Flanged	CPFL		Y

Description	Key	Plotted Isometric Shape	User-Definable (Yes/No)
Reducer - Concentric Swaged From Pipe Flanged	CSFL		Υ
Reducer - Concentric Fabricated From Plate Flanged w/ Connection	CZFL		Y
Reducer - Concentric Swaged From Pipe Flanged w/ Connection	CXFL		Y
Reducer - Concentric Nipple	RNSC		Y
Reducer - Concentric Screwed	RCSC	- I I-	Υ
Reducer - Concentric Screwed w/ Connection	CTSC	## ## ## ## ## ## ## ## ## ## ## ## ##	Y
Reducer - Concentric Screwed Bush	RBSC		Y
Reducer - Concentric Socket Weld Bush	RBSW		Y
Reducer - Eccentric Butt Weld	REBW		Y

Description	Key	Plotted Isometric Shape	User-Definable (Yes/No)
Reducer - Eccentric Fabricated From Plate	EPBW		Υ
Reducer - Eccentric Swaged From Pipe	ESBW		Υ
Reducer - Eccentric Butt Weld With a Connection	OTBW		Υ
Reducer - Eccentric Fabricated From Plate With a Connection	EZBW		Υ
Reducer - Eccentric Swaged From Pipe With a Connection	EXBW		Υ
Reducer - Eccentric Screwed	RESC	-₹-	Υ
Reducer - Eccentric Screwed With a Connection	OTSC		Υ
Reducer - Eccentric Flanged	REFL		Υ
Reducer - Eccentric Fabricated From Pipe Flanged	EPFL		Y
Reducer - Eccentric Swaged From Pipe Flanged	ESFL		Υ

Description	Key	Plotted Isometric Shape	User-Definable (Yes/No)
Reducer - Eccentric Flanged With a Connection	OTFL		Y
Reducer - Eccentric Fabricated From Plate Flanged With a Connection	EZFL		Υ
Reducer - Eccentric Swaged From Pipe Flanged With a Connection	EXFL		Y
Reducing Block	RFPL		Υ

Tees (SKEYs)

Description	Key	Plotted Isometric Shape	User-Definable (Yes/No)
Tee - Butt Weld	TEBW		N
Tee - Compression	TECP	رو اوم	N
Tee - Flanged	TEFL	го от	N
Tee - Screwed	TEFL	ہے آ دے	N
Tee - Set-on	TESO		N
Tee - Set-on Reinforced	TERF	REINFORCED	N
Tee - Socket Weld	TESW		N
Tee - Swept Branch Butt Weld	TSBW	SWEPT TEE	N

Description	Key	Plotted Isometric Shape	User-Definable (Yes/No)
Tee - Swept Branch Flanged	TSFL	SWEPT TEE	N
Tee - Swept Branch Compression	TSCP	SWEPT TEE	N Æ
Tee - Swept Branch Socket Weld	TSSW	SWEPT TEE	N E

Traps (SKEYs)

You can replace the ** characters in the symbol keys with an end prep type. For more information, see *End Prep Connections* (on page 651).

Description	Key	Plotted Isometric Shape	User-Definable (Yes/No)
Trap - In-Line (Straight Through)	TI**		Υ
Trap - Angle	TA**	4	N
Trap - Offset	TO**		N
Trap - Return	TR**		N

See Also

Valves (SKEYs)

You can replace the ** characters in the symbol keys with an end prep type. For more information, see *End Prep Connections* (on page 651).

Description	Key	Plotted Isometric Shape	User- Definable (Yes/No)
Valve - Angle	AV**	F	Υ
Valve - Angle Pressure Reducing	AX**		Y
Valve - Angle Relief/Vent	AR**	*	Υ
Valve - Ball	VB**		Υ
Valve -Basic	VV**		Υ
Valve - Butterfly	VY**		Υ
Valve - Check	VC**		Υ
Valve - Check	CK**		Υ
Valve - Cock	VK**		Y

Description	Key	Plotted Isometric Shape	User- Definable (Yes/No)
Valve - Diaphragm	VD**		Y
Valve - Gate	VT**		Y
Valve - Globe	VG**		Y
Valve - Needle	VN**		Υ
Valve - Plug	VP**		Υ
Valve - Pressure Reducing	VX**		Y
Valve - Relief/Vent	VR**	*	Y
Valve - Slide	VS**		Y
		r	

Three-Way Valves

Description	Key	Plotted Isometric Shape	User- Definable (Yes/No)
Valve - 3-Way	V3**	1	Y

Four-Way Valves

Description	Key	Plotted Isometric Shape	User- Definable (Yes/No)
Valve - 4-Way	V4**		Υ

See Also

Appendix: Symbols and Symbol Keys (on page 552)

Vents (SKEYs)

Description	Key	Plotted Isometric Shape	User- Definable (Yes/No)
Rupture Disk	RD		Υ

See Also

Welds (SKEYs)

Description	Key	Plotted Isometric Shape	User- Definable (Yes/No)
Weld - Site	WS	\star	Υ
Weld - Field Fitted	WF	\star	Υ
Weld - Workshop	WW	•	Υ
Weld Mitre (Shop)	WM	MITRE 90.0°	<i>κ</i> ^N
Weld Mitre (Site)	WM	MITRE 90.0°	K
Weld Mitre (Offshore)	WM	MITRE 90.0°	<i>κ</i> ^N
Weld Mitre (Field Fit)	WMF	FFW MITRE 90.0°	N
Weld Mitre (Field Fit)	WMF	FFW MITRE 90.0°	N
Offshore Weld	WO	*	Υ
Offshore Weld-Field Fit	WOF	*	Υ

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